GMS User Manual (v9.1)

The Groundwater Modeling System

PDF generated using the open source mwlib toolkit. See http://code.pediapress.com/ for more information.
PDF generated at: Wed, 08 May 2013 16:51:22 UTC

Contents

Articles

1. Learning GMS **1**

What is GMS? 1

The GMS Screen 3

Tool Palettes 6

Project Explorer 9

Tutorials 10

2. Set Up **11**

64 bit 11

License Agreement 11

Registering GMS 12

Community Edition 13

Graphics Card Troubleshooting 14

Reporting Bugs 18

3. General Tools **19**

3.1. The File Menu **20**

The File Menu 20

3.2. The Edit Menu **22**

The Edit Menu 22

Units 24

Preferences 24

Materials 29

Material Set 29

3.2.1. Coordinate Systems **31**

Coordinate Systems 31

Coordinate Conversions 32

Projections 33

CPP Coordinate System 35

Geographic Coordinate System 35

Local Coordinate System 36

Coordinate Transformation 36
Transform 37

3.3. The Display Menu **39**

The Display Menu 39
Contour Options 41
Animations 44
Color Ramp 46

3.3.1. Display Options **48**

Display Options 48
Drawing Grid Options 49
Vectors 50
Lighting Options 51
Plot Axes 53

3.4. Other Tools **54**

Annotations 54
CAD Options 58
Cross Sections 59
Data Sets 63
Data Calculator 66
Display Theme 68
XY Series Editor 69

4. Interpolation **70**

4.1. Introduction **71**

Interpolation 71
Interpolation Commands 72
3D Interpolation Options 73
Steady State vs. Transient Interpolation 74

4.2. Linear **75**

Linear 75

4.3. Inverse Distance Weighted **76**

Inverse Distance Weighted 76
Shepards Method 77
Gradient Plane Nodal Functions 78

Quadratic Nodal Functions 79
Subset Definition 80
Computation of Interpolation Weights 81

4.4. Clough-Tocher **83**
Clough-Tocher 83

4.5. Natural Neighbor **84**
Natural Neighbor 84

4.6. Kriging **87**
Kriging 87
Kriging Options 90
3D Kriging 91
Variogram Editor 92

4.7. Jackknifing **99**
Jackknifing 99

5. Modules **100**

5.1. TIN Module **101**
TIN Module 101
Creating a TIN 102
Editing a TIN 104
TIN Settings 108
TIN Display Options 109
TIN Tool Palette 112
Converting TINS to Other Data Types 114
Building Solids and 3D Meshes with TINs 116
Triangulation 118
TIN Files 119

5.2. Boreholes Module **123**
Boreholes Module 123
Creating and Editing Boreholes 125
Borehole Display Options 127
Borehole Tool Palette 129
Borehole Hydrogeologic Units 131
Converting Borehole Data 133

Borehole Cross Sections 134

5.2.1. Horizons **138**

Horizons 138

Horizon Conceptual Model 140

Horizons Applications 142

Horizons to HUF 145

Horizons Wizard 147

Horizons to Solids 148

Horizons to 3D Mesh 150

5.3. Solid Module **152**

Solid Module 152

Solid Properties 154

Solid Primitives 156

Solid Display Options 157

Solid Module Tool Palette 158

Solids to Layered Mesh 159

Solids to HUF 160

Solids to MODFLOW Command 161

5.4. 2D Mesh Module **168**

2D Mesh Module 168

Creating a 2D Mesh 170

Editing 2D Meshes 172

2D Mesh Settings 176

2D Mesh Display Options 177

2D Mesh Tool Palette 179

Converting a 2D Mesh to other types of Data 182

Element types 184

2D Mesh Polygon Attributes 186

5.5. 2D Grid Module **190**

2D Grid Module 190

2D Grid Types 192

Creating and Editing 2D Grids 193

2D Grid Display Options 195

2D Grid Tool Palette 197

Converting 2D Grids 199

5.6. 2D Scatter Point Module **200**
2D Scatter Point Module 200
Creating and Editing 2D Scatter Point Sets 202
2D Scatter Point Display Options 204
2D Scatter Point Tool Palette 205
Interpolating with 2D Scatter Points 206
Converting 2D Scatter Points to Other Types of Data 207
Gaussian Field Generator 208
Active/Inactive Points 212
2D Interpolation Options 213
5.7. 3D Mesh Module **215**
3D Mesh Module 215
3D Element Types 216
Creating a 3D Mesh 217
Editing a 3D Mesh 220
3D Mesh Display Options 224
3D Mesh Tool Palette 226
Classify Material Zones 228
Iso Surfaces 229
Converting 3D Meshes to Other Data Types 232
Building the 3D Mesh from the FEMWATER Conceptual Model 234
5.8. 3D Grid Module **236**
3D Grid Module 236
3D Grid Types 238
Creating and Editing 3D Grids 239
3D Grid Display Options 241
3D Grid Tool Palette 242
Classify Material Zones 243
3D Grid Viewing Modes 244
Converting 3D Grids to Other Data Types 246
Exporting Grids 248
Cell Properties 250
Active/Inactive Cells 251
Named Layer Ranges 252
Redistribute Grid Cells 253

Redistribute Layers 254

5.9. 3D Scatter Point Module **255**

3D Scatter Point Module 255

3D Scatter Point Display Options 256

3D Scatter Point Tool Palette 257

Interpolating with 3D Scatter Points 258

Converting 3D Scatter Points to Other Data Types 259

Bounding Grid 260

5.10. Map Module **261**

Map Module 261

Feature Objects 262

Conceptual Model 267

Feature Object Display Options 268

Feature Object Tool Palette 270

Coverages 272

Grid Frame 275

Clean Command 276

Temporal Discretization 277

Map to Models 279

Map to Modules 280

5.11. GIS Module **283**

GIS Module 283

GIS Display Options 284

GIS Tool Palette 285

Enabling ArcObjects 286

GIS to Feature Objects 287

Add Data 288

Arc Hydro Groundwater 289

6. Models **290**

6.1. FEMWATER **291**

FEMWATER 291

6.1.1. FEMWATER Pre-Processing **293**

Building a FEMWATER Model 293

FEMWATER Conceptual Model Approach 297
FEMWATER Flows 300
Point Source/Sink BC 301
Assign Node/Face BC 302
FEMWATER Point / Arc Properties 303
FEMWATER Model Input 306
FEMWATER Initial Conditions 309
Saving a FEMWATER Simulation 312

6.1.2. FEMWATER Post-Processing **313**
FEMWATER Display Options 313
FEMWATER Post Processing Viewing Options 315
FEMWATER Particle Sets 316

6.2. MODAEM **317**
MODAEM 317
MODAEM License Agreement 318
MODAEM Display Options 325

6.3. MODFLOW **326**
MODFLOW 326
MODFLOW 2005 329

6.3.1. MODFLOW Packages **332**
MODFLOW Packages Supported in GMS 332
BCF Package 337
CHD Package 340
DE4 Package 342
DRN Package 343
DRT Package 345
ETS Package 347
EVT Package 350
GHB Package 353
Global Options/Basic Package 355
GMG Package 359
HFB Package 360
HUF Package 363
LAK Package 365
LMG Package 366

LMG1 Package 369
LPF Package 371
MNW1 Package 374
MNW2 Package 376
NWT Package 378
PCG2 Package 379
RCH Package 381
PEST ASP Package 383
RIV Package 386
SIP Package 388
SSOR Package 390
STR/SFR Packages 392
UZF Package 395
WEL Package 397

6.3.2. MODFLOW Pre-Processing **399**

Building a MODFLOW Model 399
MODFLOW Array Editor 401
MODFLOW Grid Approach 403
MODFLOW Conceptual Model Approach 405
Activate Cells in Coverage 409
Stress Periods 410
Map to MODFLOW 412
Defining the Layer Elevations 416
Standard MODFLOW Parameters 419
MODFLOW Parameters - Two Methods 427
Conductance 428
MODFLOW Parameters Disclaimer 431
Saving and Reading a MODFLOW Simulation 432
Output Control 433
Importing MODFLOW Files 435
MODFLOW with HDF5 443
Unsupported MODFLOW Features 447
Interpolate to MODFLOW Layers 449
MODFLOW Layers to 2D scatter points 451
MODFLOW Source/Sink Dialog 453

6.3.3. MODFLOW Post-Processing **456**

MODFLOW Display Options 456
MODFLOW Post Processing Viewing Options 458
MODFLOW Solution Properties Dialog 461
Saving and Reading a MODFLOW Simulation 464
Cell Properties 466
MODFLOW World File 467
Error Summary Plot 468
Zone Budget 470
ZONEBUDGET 473
Viewing Computed Fluxes 474

6.4. MODPATH **475**

MODPATH 475
MODPATH Particle Tracking 476
MODPATH Particle Sets 478
MODPATH Zone Codes 479
MODPATH Display Options 480
Exporting Pathlines 481

6.5. MT3DMS **483**

MT3DMS 483
MT3D Packages 484
MT3D Source/Sink Mixing Package 485
Basic Transport Package Dialog 486
ICBUND Array 489
MT3D Source/Sink Mixing Package 490
MT3D point Sources/Sinks BC 492
Building an MT3DMS Simulation 493
Saving an MT3DMS Simulation 494
Importing an MT3D Simulation 494
MT3DMS Display Options 495
Calculate Mass 495

6.6. PEST **496**

PEST 496

6.7. PHT3D **498**

PHT3D 498

6.8. RT3D **500**
RT3D 500
Basic Transport Package Dialog 502
6.9. SEAM3D **505**
SEAM3D 505
6.10. SEAWAT **506**
SEAWAT 506
Building a SEAWAT Model 507
SEAWAT Global Options Dialog 508
SEAWAT MODFLOW AUX Variables 509
SEAWAT Packages 510
SEAWAT Importing a SEAWAT Simulation 512
6.11. SEEP2D **513**
SEEP2D 513
SEEP2D Display Options 517
SEEP2D Solution 518
6.12. T-PROGS **519**
T-PROGS 519
Vertical Markov Chain 526
Strike Dip Markov Chain 531
TSIM 533
6.13. UTEXAS **535**
UTEXAS 535
UTEXAS Display Options 541
6.14. WASH123D **542**
WASH123D Display Options 542
7. Modeling **544**
7.1. Calibration **545**
Model Calibration 545
Automated Parameter Estimation 547
Parameter Estimation Dialog 550

PEST 556
Run Options 558
Observations 562
Plot Wizard 566
Calibration Targets 579
Model Checker 580

7.2. Parameters **581**
Parameters 581
Parameter Dialog 585
Pilot Points 590
Multiplier Arrays for Parameters 593
Standard MODFLOW Parameters 594

7.3. Stochastic Modeling **602**
Stochastic Modeling 602
Gaussian Field Generator 607
Risk Analysis Wizard 610

8. Importing/Exporting Data **616**
Importing Non-native GMS Files 616
Text Import Wizard 617
Text Import Wizard Supported File Formats 618
Importing/Exporting CAD Data 625
Importing USGS DEMs 626
Importing/Exporting GIS Grids 627
Importing/Exporting Shapefiles 628
Shapefiles 632
Import From Database 632
Get Online Maps 633
Exporting Non-native GMS Files 634
Images 635
Registering an Image 637
KMZ files 639
GeoStudio Import 641

9. Appendix **642**
File Formats 642
Data Set Files 643

Bugfixes GMS 652
Dialog Help 674

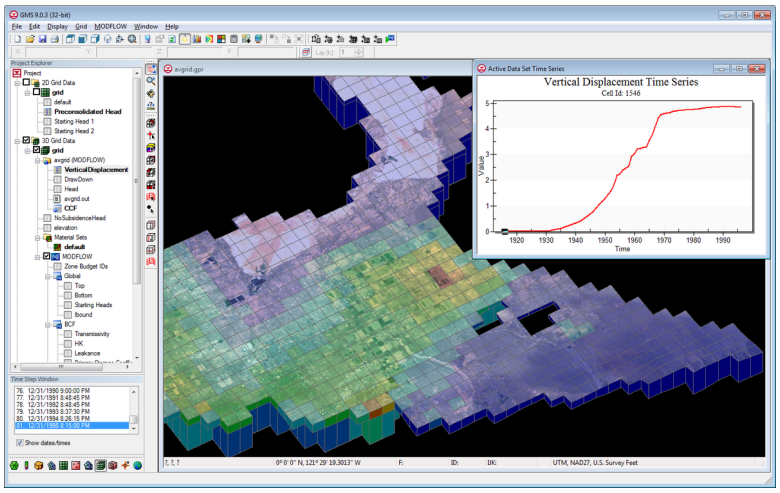
References

Article Sources and Contributors 684
Image Sources, Licenses and Contributors 691

1. Learning GMS

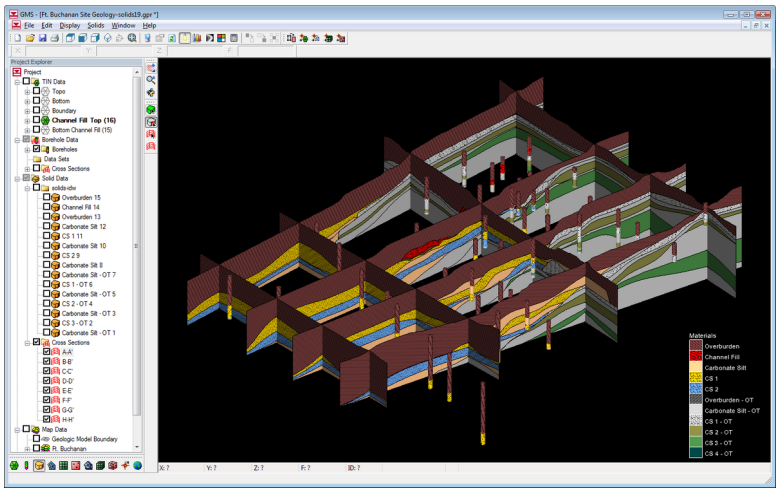
What is GMS?

The Groundwater Modeling System (GMS) is a comprehensive graphical user environment for performing groundwater simulations. The entire GMS system consists of a graphical user interface (the GMS program) and a number of analysis codes (MODFLOW, MT3DMS, etc.). The GMS interface is developed by Aquaveo, LLC in Provo, Utah.



SUB package vertical displacement plot.

GMS was designed as a comprehensive modeling environment. Several types of models are supported and facilities are provided to share information between different models and data types. Tools are provided for site characterization, model conceptualization, mesh and grid generation, geostatistics, and post-processing.



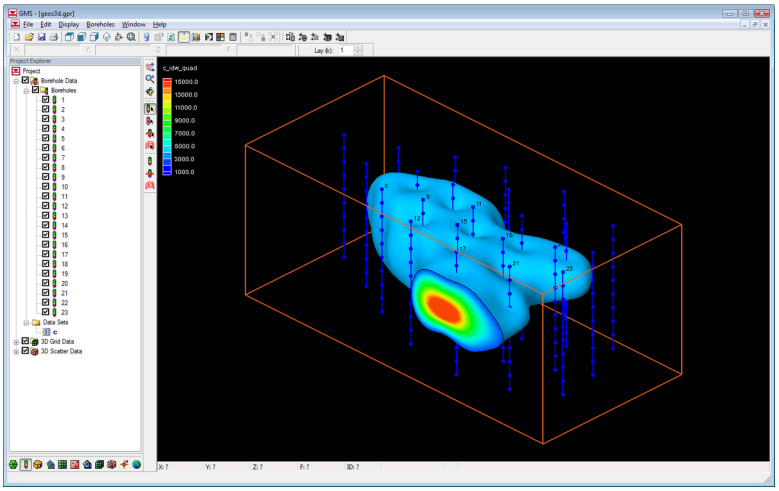
Solid cross sections and boreholes.

Modules

The interface for GMS is divided into eleven modules. A module is provided for each of the basic data types supported by GMS. As you switch from one module to another module, the**Dynamic Tool Palette** and the **Menus** change. This allows you to focus only on the tools and commands related to the data type you wish to use in the modeling process. Switching from one module to another can be done instantaneously to facilitate the simultaneous use of several data types when necessary. The following modules are supported in GMS:

TIN Module

- Borehole Module
- Solid Module
- 2D Mesh Module
- 2D Grid Module
- 2D Scatter Point Module
- 3D Mesh Module
- 3D Grid Module
- 3D Scatter Point Module



Isosurface on a 3D grid and borehole sample data.

- Map Module
- GIS Module

Numerical Models

Numerical models are programs that are separate from GMS that are used to run an analysis on a model. The models can be built in GMS, and then run through the numerical model program. GMS can then read in and display the results of the analysis.

With most numerical models, GMS has the option of using a Model Wrapper to run the model and display real-time results during the model simulation.

The following numerical models are currently supported in GMS.

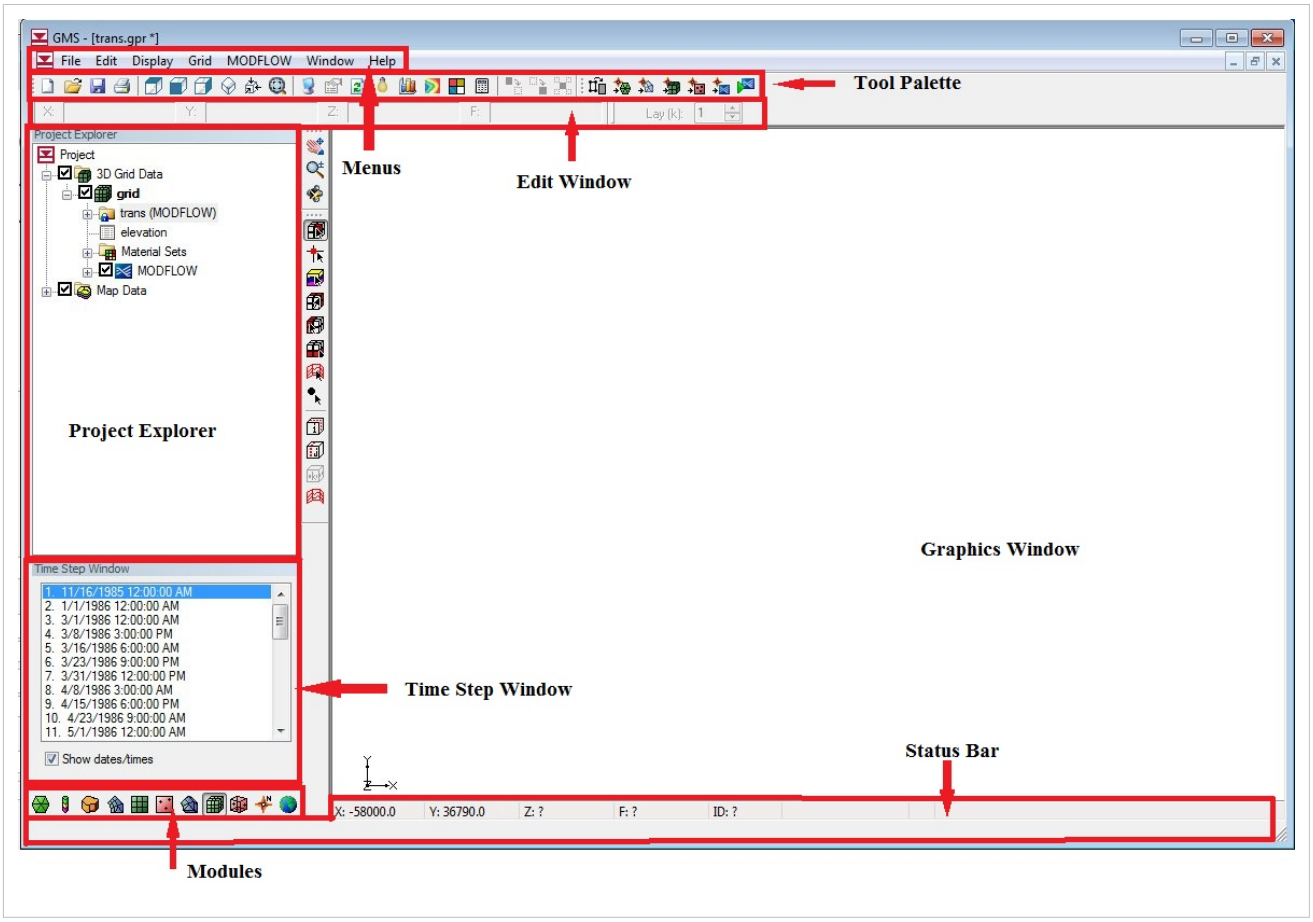
Model Name	GMS Module
• MODFLOW • MODPATH • MT3DMS • RT3D • ART3D • SEAM3D • UTCHEM • PEST • SEAWAT (starting v8.0)	3D Grid
• MODAEM • UTEXAS	Map Module
• SEEP2D	2D Mesh
• FEMWATER	3D Mesh
• T-PROGS	Boreholes

Source code for most models is available upon request. Contact technical support [1] to request source code.

References

[1] http://www.aquaveo.com/technical-support/

The GMS Screen



The GMS screen is divided into seven main sections:

Menu Bar

Each module has its own set of menus. The first three menus (File, Edit, Display) are the same for every module. The remaining menus are dependent on the selected module.

Edit Window

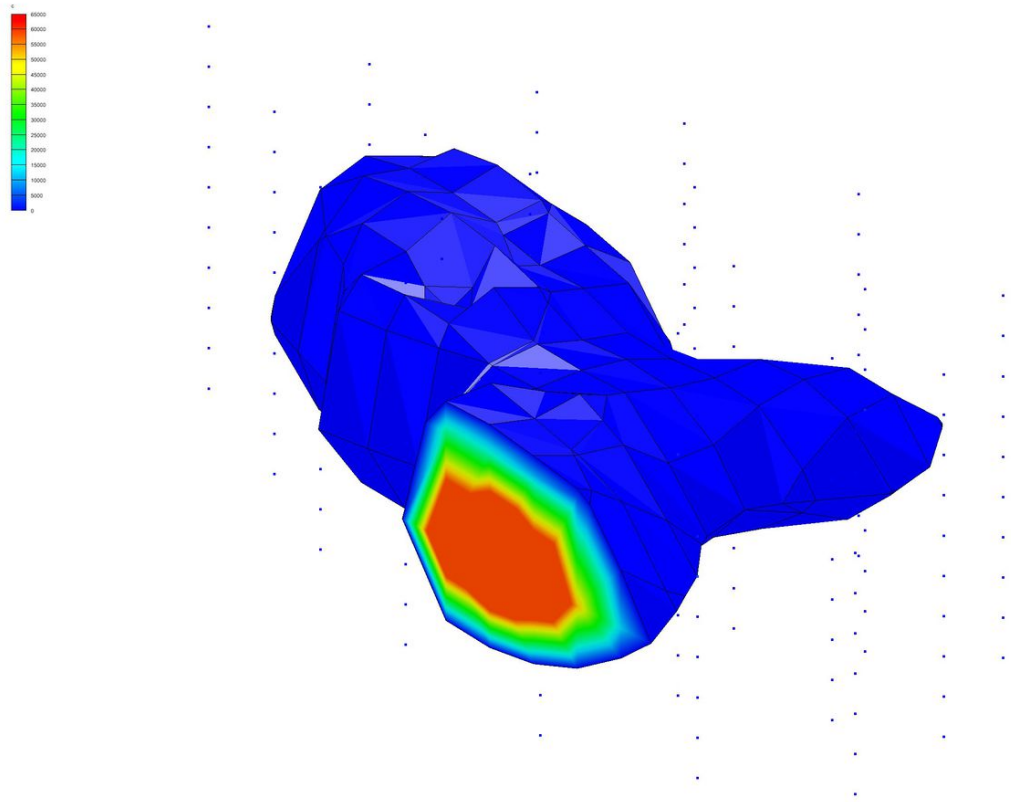
These fields are used to edit the coordinates of selected items (vertices, nodes, scatter points, etc.). The coordinates are changed by typing in new values and hitting the ENTER or TAB key. The scalar data value (F:) associated with the selected object can also be edited.

Tool Palette

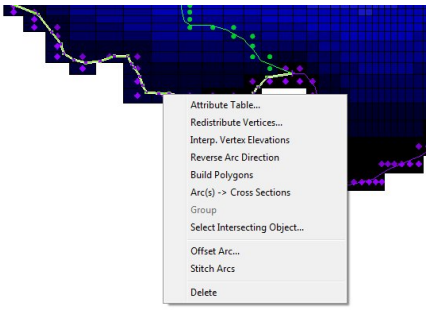
Several Tool Palettes can be displayed in the GMS interface. Default tool palettes include the Macros Palette, the Static Tool Palette, the Dynamic Tool Palette, the Mini-Grid Plot Palette, and the Module Palette. (For more information on each of these palettes see Tool Palettes)

Graphics Window

The primary graphical input and output for GMS takes place in the Graphics Window. The action taken when you interact with the Graphics Window depends on which tool is selected.



Every object that can be selected in the Graphics Window now has a corresponding right-click menu available. These menus contain commands that allow the user to manipulate the data or display of the selected items.



Example of a right-click menu on a arc in the map module.

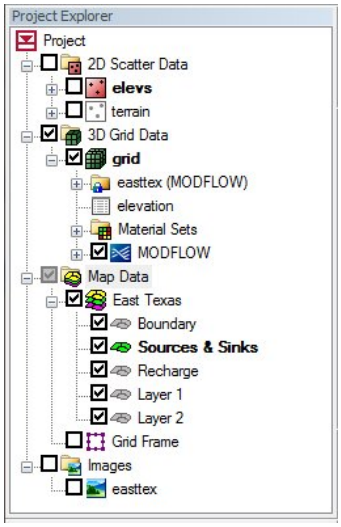
Project Explorer

The Project Explorer is located at the left side of the GMS window by default, but it can be moved to anywhere on the window since it is a dockable toolbar. The Project Explorer contains a hierarchical representation of the data associated with a modeling project. The Project Explorer can also be resized simply by clicking on the window borders and dragging them to a new location.

Many commands on the data in GMS can be executed by right-clicks in the Project Explorer. For general commands or to create new data objects you can right-click on the empty space in the Project Explorer and the following pop-up menu is available. The visibility of items in the Graphics Window can also be controlled by selecting the toggle next to each item in the Project Explorer.

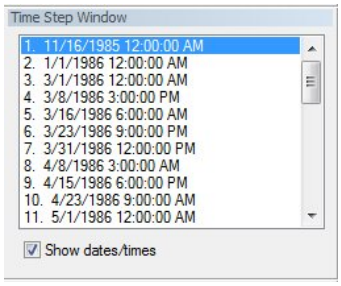
Click here to learn about data sets in the Project Explorer.

Project Explorer View



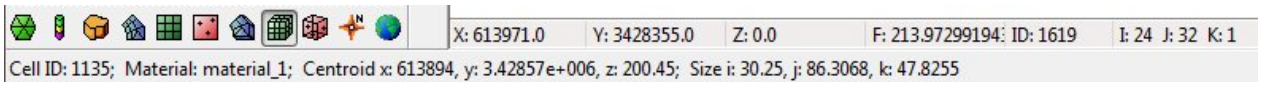
Time Step Window

The Time Step Window is located below the Project Explorer by default, but it can be moved to anywhere on the window since it is a "dockable" toolbar. The Time Step Window only appears when a transient data set is selected in the Project Explorer. Time steps can be viewed in date/time format or relative time format by selecting the toggle below the Time Step Window. The Time Step Window can be resized simply by clicking on the window borders and dragging them to a new location.



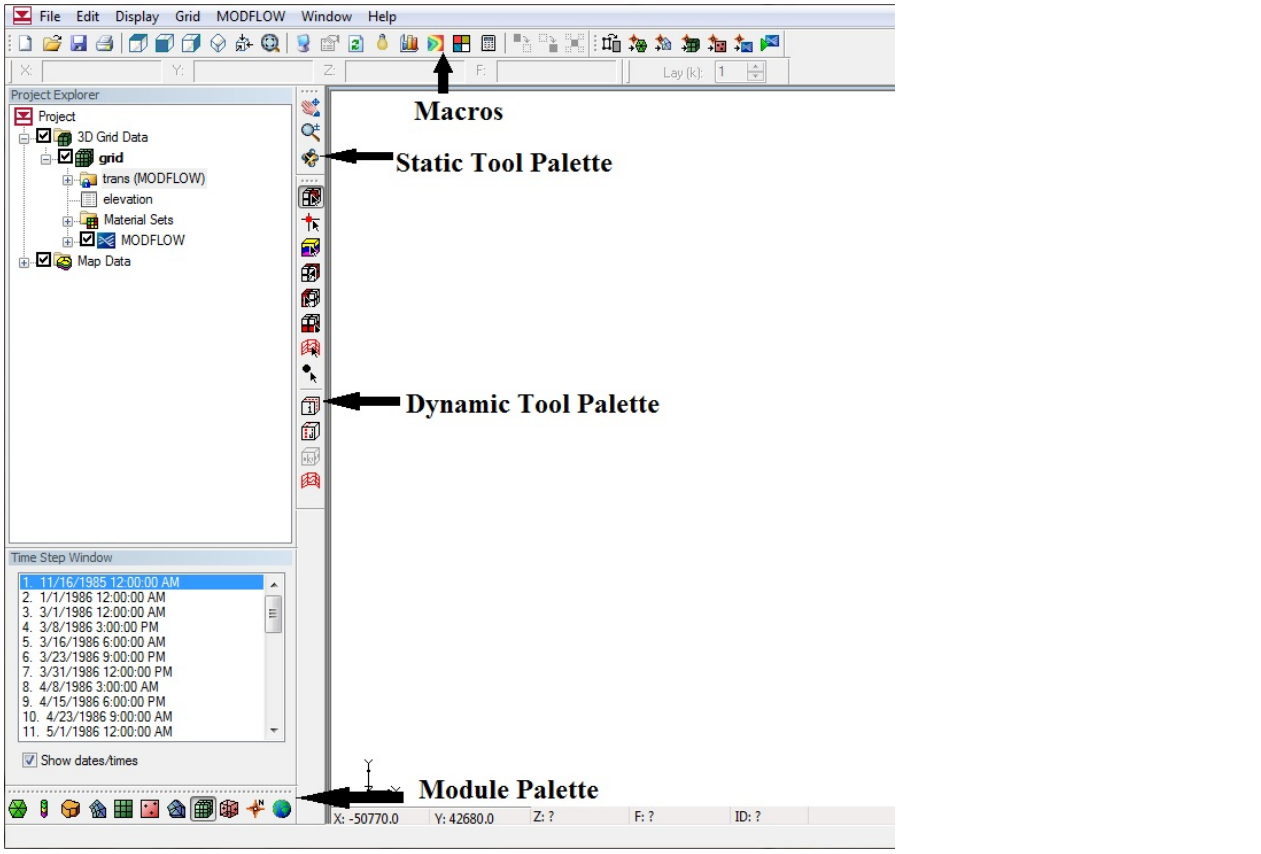
Status Bar

The Status Bar is used to display the cursor coordinates, the IJK indices of the cell beneath the cursor, the data set value beneath the cursor, data associated to selected items, and other information.



Tool Palettes

There are several Tool Palettes that can be displayed in the GMS interface. Below are the tool palettes that are on by default.



Macros

Many of the more frequently used menu commands can be accessed through the macro buttons. These buttons essentially serve as shortcuts to menu commands.

Static Tool Palette

The Static Tool palette contains the tools which are available in every module. These tools are tools for basic operations such as panning and zooming. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the Static Tool palette.

Tool	Tool Name	Description
	Pan	The Pan tool is used to pan the viewing area of the Graphics Window. Panning can be done in 3 ways: • When the Pan tool is active, holding down the main mouse button while dragging moves the view. • If another tool is active and you don't want to switch tools, you can pan by holding down the F2 key and clicking and dragging with the mouse. • If your mouse has a middle button (or a mouse wheel), you can hold it down and drag to pan the view.
	Zoom	The viewing area can be magnified/shrunk using the Zoom tool. Zooming can be done in the following ways: • With the zoom tool selected, clicking on the screen zooms the display in around the point by a factor of two. Holding down the shift key zooms out. • With the zoom tool selected, a rectangle can be dragged around a portion of the display to zoom in on that region. Holding down the shift key zooms out. • If another tool is active and you don't want to switch tools, you can zoom by holding down the F3 key and clicking and dragging with the mouse. • If your mouse has a middle button (or a mouse wheel), you can scroll the wheel to zoom in and out.
	Rotate	The Rotate tool provides a quick way to rotate the viewing location. Rotating can be done in the following ways: • With the rotate tool selected, holding down the mouse button and dragging the cursor in the Graphics Window rotates the object in the direction specified. A horizontal movement rotates the image about the z axis. A vertical movement rotates the image about the x and y axis. The amount of rotation depends on the length the cursor moves while the mouse button is down. • If another tool is active and you don't want to switch tools, you can rotate by holding down the F4 key and clicking and dragging with the mouse. • The viewing angle can also be entered directly.
	Measure	The Measure tool provides a quick way to measure distances. • The tool is only available in plan view. • Measuring is done by selecting the tool and clicking on the Graphics Window with the mouse. A single line or a polyline can be created with a double-click used to end the line. The backspace key removes the last point clicked. • The instantaneous and total lengths are given at the bottom of the GMS window. The length units correspond to those selected in the Units dialog. A units conversion takes place if the current horizontal Projection units are different than those in the Units dialog.

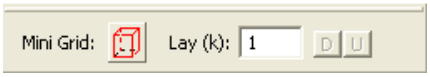
Dynamic Tool Palette

When the active module is changed by selecting an object in the Project Explorer, the tools in the Dynamic Tool Palette change to the set of tools associated with the selected object/module. Each module has a separate set of tools.

3D Grid Tools - This is just an example of one of the Dynamic Tool Palettes

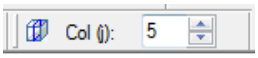
Mini-Grid Plot

The Mini-Grid Plot appears when a 3D grid is in memory and is activated when the orthogonal viewing mode is active. In the orthogonal mode, the viewing angle is always parallel to one of the three grid axes (I, J, or K) and only one of the rows, columns, or layers is displayed at one time. The Mini-Grid Plot shows an idealized representation of the 3D grid and shows which of the rows, columns, or layers is currently being displayed. The current row, column, or layer can be changed using the arrows just below the Mini-Grid Plot.



Mini-Grid Toolbar

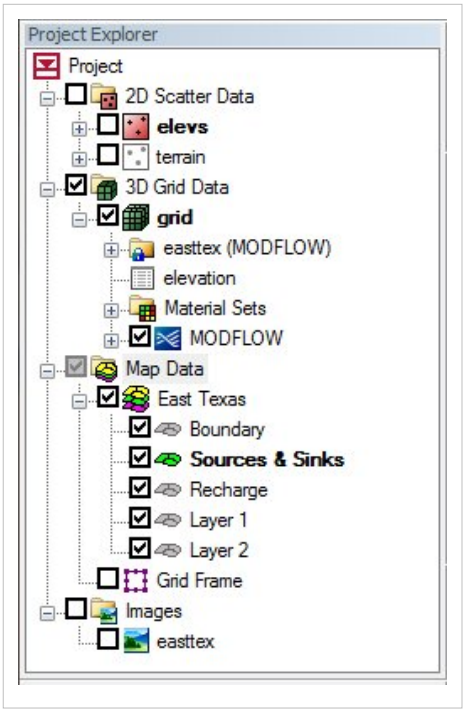
The Mini-Grid Toolbar appears when a 3D grid is in memory and is activated when the orthogonal viewing mode is active. In the orthogonal mode, the viewing angle is always parallel to one of the three grid axes (I, J, or K) and only one of the rows, columns, or layers is displayed at one time. The Mini-Grid Toolbar shows an idealized representation of the 3D grid and shows which of the rows, columns, or layers is currently being displayed. The current row, column, or layer can be changed using the arrows just below the Mini-Grid Toolbar.



Module Palette

The Module Palette is used to switch between modules. Only one module is active at any given time. However, the data associated with a module (ex. a 3D finite element mesh) is preserved when the user switches to a different module. Activating a module simply changes the set of available tools and menu commands.

Project Explorer



The Project Explorer is located at the left side of the GMS window by default. It can be moved to anywhere on the window since it is a "dockable" toolbar. The Project Explorer contains a hierarchical representation of the data associated with a modeling project. Previously only the data form the active module was displayed in the Project Explorer, but now all of the data in GMS is always displayed independent of the active module. The new Project Explorer can also be resized simply by clicking on the window borders and dragging them to a new location.

All of the modules have root items coresponding to them. Each root item may be right clicked on to accomplish certain actions. Data for each module is grouped into folders. The modules and items can also be expanded and colapsed to show sub folders. Each item may also be turned on or off by clicking the check box next to the item. Many items in the project explorer can be dragged to different locations to be used with different modules. Items may also be duplicated.

Many commands on the data in GMS can be executed by right-clicks in the Project Explorer. For general commands or to create new data objects you can right-click on the empty space in the Project Explorer and the following pop-up menu is available. Right click commands can also be used to export or transform items. The visibility of items in the Graphics Window can also be controlled by selecting the toggle next to each item in the Project Explorer.

Tutorials

A rich set of step-by-step tutorials has been developed to aid in learning how to use GMS.

The tutorials are in PDF format. They are installed in the "docs" or "tutfiles" directory in the folder where GMS is installed. They can also be accessed by clicking on the links below.

The tutorials are listed below by subject and are **not** necessarily listed in the suggested order of completion. Some tutorials assume a basic knowledge of GMS and some build on other tutorials. When this is the case the tutorial itself will state which tutorials you should complete beforehand.

Many of the tutorials have files that are needed to run the tutorial. These files can be found in the "tutfiles" folder where GMS was installed. They can also be downloaded here.

NOTE: In GMS 7.1 and up, the default location for the tutorials is the users "Documents" (Windows Vista and 7) or "My Documents" (Windows XP) folder.

[1].

Previous Versions

For previous versions of the GMS tutorials, click on one of the following links. For the current version, see the tables below.

- GMS 6.5 Tutorials
- GMS 7.0 Tutorials
- GMS 7.1 Tutorials
- GMS 8.0 Tutorials
- GMS:Tutorial Tables 8.1 [2]
- GMS:Tutorial Tables 8.2 [3]
- GMS:Tutorial Tables 8.3 [4]

References

[1] http://www.aquaveo.com/gms-learning
[2] http://www.aquaveo.com/gms-81tutorials
[3] http://www.aquaveo.com/gms-82tutorials
[4] http://www.aquaveo.com/gms-83tutorials

2. Set Up

64 bit

Starting at GMS 8.1, GMS is available in a 64 bit version. This means:

- GMS can access more RAM so larger models can be created
- A 64 bit version of MODFLOW is included and can be used by changing an option in the Preferences dialog
- ArcObjects is not available because ESRI [1] has not created a 64 bit version

References

[1] http://www.esri.com

License Agreement

AQUAVEO SOFTWARE LICENSE AGREEMENT

We understand that neither Aquaveo LLC nor its employees makes any warranty express or implied, or assumes any legal responsibility for the accuracy, completeness, or usefulness of the computer programs and documents herein ordered. We acknowledge that Aquaveo's liability for damages associated with these programs is limited to the amounts paid to Aquaveo. We understand that we may register the executable on only one machine per license purchased. We also agree to not make the program available to more persons, at any given time, than the number of licenses purchased. We agree to not distribute this program in unmodified or modified form, outside our organization without the written permission of Aquaveo.

License agreements for software used by Aquaveo software

FTGL LICENSE AGREEMENT

http://ftgl.wiki.sourceforge.net/states: "FTGL is free software. You may use it, modify it and redistribute it under the terms of the MIT license or the GNU LGPL, at your option."

The MIT license link is to http://en.wikipedia.org/wiki/MIT_License, which states:

Copyright (c) <year> <copyright holders> Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION

WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Registering GMS

When you first install GMS, it will be running in Demo Mode. All GMS functions will be enabled with the exception of printing and saving. Anyone can run GMS in Demo Mode on any computer and it can be freely distributed. To enable the print and save functions, you need either a password or a hardware lock.

The components (modules, interfaces) can be licensed individually depending on the needs and interests of the user. The components of GMS are licensed using a password system. The ***Register*** command is used to enter a password that enables the licensed components. This command can be used to enable the program after initially installing GMS, or for adding additional modules to the program at a later time. The ***Register*** command must be used before any files can be saved or printed. Before registration, GMS will run in Demo Mode.

When the ***Register*** command is selected, the *Register* dialog appears. The first item shown in the dialog is the security string. This string is keyed to the hard drive of the computer where GMS is installed and uniquely identifies the computer. When first registering GMS, this security string should be reported to the distributor or reseller where GMS was purchased. The reseller then provides a password which should be entered in the edit field at the top of the dialog. Once the password is entered, the **Register** button is selected. If the security string was reported correctly and the password was entered correctly, the text next to each of the licensed components changes from "DISABLED" to "ENABLED".

The **Details** button brings up a dialog listing the phonetic code for the security string. When reporting the security string over the phone to get a password, using the phonetic code can be helpful in avoiding errors.

Once GMS has been registered, a file called gmspass.txt is created. Since GMS is licensed on a per/seat basis, arrangement must be made to get an additional password if GMS is to be moved to another computer.

Also, GMS can also be enabled using a hardware lock, rather that a password. Contact your GMS reseller for details.

Password

From the *File* menu, select the ***Register*** command. This brings up the Register dialog, which has a "security string" listed at the top. Send this security string along with your name, company, phone number, and e-mail address to your vendor.

After verifying that you are a licensed user, a password will be sent to you. When you receive the password, enter it in the **Password** field in the *Register* dialog and click the **Register** button. The modules you purchased will become enabled and GMS will run in Normal mode.

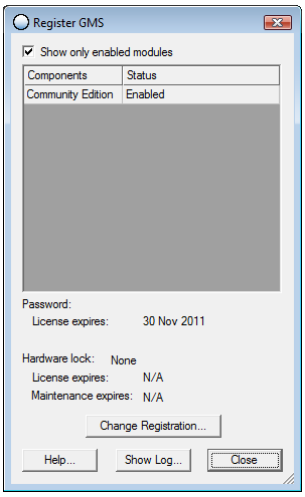
Hardware Lock

Follow the instructions you received with the hardware lock to install the hardware lock and accompanying drivers. If you did not receive hardware lock instructions, or they have been misplaced, they can be found in the \Utils\Hwlock\Instructions directory on the CD. There are separate files for single user and network hardware locks. These files can be read using your web browser. If you would like to purchase or have questions about hardware locks, please contact your vendor.

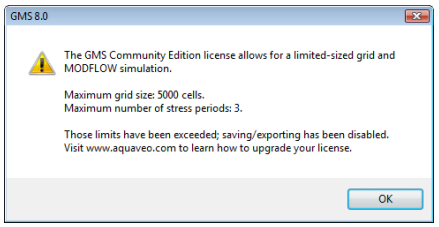
Community Edition

Starting at version 8.0 there is a free version of GMS called "Community Edition". It is limited to include only the 3D grid module and the MODFLOW model interface. It is also restricted in the size of the grid and the number of MODFLOW stress periods. Any size model can be imported, but if the grid exceeds 5000 cells or the number of stress periods is more than 3 the project cannot be saved and a watermark is displayed in the graphics window. The community edition must still be registered using a license code which can be obtained via the internet from the Registration Wizard (Help | Register | Change Registration | Get Community Edition License).

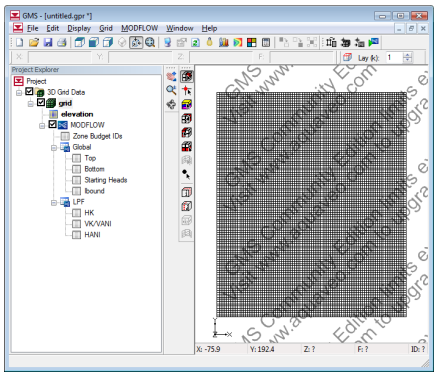
You can check if you are running in Community Edition mode by going to the Registration dialog. The size limits are displayed in the Help|About dialog.



GMS registration dialog showing that GMS is running in Community Edition mode.



GMS Community Edition warning dialog.



GMS showing watermark in Community Edition when size limits are exceeded.

Graphics Card Troubleshooting

XMS (WMS, GMS, or SMS) use OpenGL for rendering graphics. OpenGL is a graphics standard, but each implementation is maintained by individual graphics card companies. Different graphics cards and drivers support different versions of the OpenGL standard. XMS currently uses features up to version 1.5 of OpenGL (as of April 2009 version 3.1 was most recent version).

Some graphics cards, as well as remote desktop, do not support functionality through OpenGL version 1.5. This is mostly a problem with older integrated graphics cards, in particular those manufactured by Intel. This page will give you some ideas on troubleshooting these problems. The best solution is to get a graphics card that supports later versions of OpenGL. You will see improved performance as well as be able to access all the features of XMS.

Remote Desktop

XMS (WMS, GMS, or SMS) will have reduced capability when running remote desktop.

Since remote desktop only supports OpenGL version 1.1 not all of the features of XMS may be available.

1. One solution is to use a different remote control software that utilizes the graphics card of the computer you are controlling. www.logmein.com [1] has free and paid versions of remote desktop that behave better with XMS. RealVNC is a program that does this and can be purchased at a reasonable cost. There is a free version but it has not been tested with the XMS software. See VNC Homepage [2] for more information.
2. Another solution is to use the Mesa software rendering option available in the application's graphic preferences. See the section below on OpenGL Graphics Dialogs for discussion of this option.

Parallels Desktop for Mac

XMS has reduced capability when running in a pure virtual PC through Parallels Desktop for Mac. Although Parallels version 6.0 provides OpenGL version 2.1 support (instead of OpenGL version 1.1) when "Enable 3D acceleration" is selected in the virtual machine's hardware configuration, the Parallels virtual video card adapter does not render all XMS graphics correctly. The solution is to use the Mesa software rendering option available in XMS's graphic preferences. See the section below on OpenGL Graphics Dialogs for discussion of this option.

If you are running XMS in a virtual PC utilizing a Boot Camp partition then Parallels uses the actual graphics card installed in the Mac. See sections below regarding graphics card issues.

OpenGL Graphics Dialogs

XMS (post WMS 8.2, GMS 7.0 onward, and SMS 10.1 onward) have dialogs that allow the selection of OpenGL support. The choice is between the system default library and the Mesa software library. The system default can change based upon current conditions such as a remote login. Not all system defaults support all needed graphics functionality. Therefore Mesa is provided for better functionality at a potential reduction in speed. However, Mesa may produce poor images when printing. The user can make this tradeoff in the graphics dialog found in preferences. The dialog provides 4 options so that on subsequent runs XMS will:

1. Ask which graphics library to use if the system does not support all OpenGL functionality needed by XMS. This option is initially set and gives the following options:
    1. Autoselect the Mesa software library for this run if the system default does not support all functionality. XMS will not prompt on subsequent runs. It will just check support and select a library.
    2. Use the system default library on this run (and on future runs if the "Do not ask again box" is checked).
    3. Use the Mesa software library on this run (and on future runs if the "Do not ask again box" is checked).
2. Autoselect the Mesa software library if the system default does not support all functionality.

3. Always use the system default library.
4. Always use the Mesa software library.

Determining Graphics Card Manufacturer

Always download and install the latest drivers from your graphics card vendor. Graphics card problems are often due to using the wrong or outdated drivers. You can use a simple diagnostic program called dxdiag [3] to determine your computer's hardware, operating system, and graphics card. To use the dxdiag [3] program:

1. Select "Start"
2. Choose "Run."
3. Type "dxdiag" in the box and click "OK."
4. Click "Yes" to the prompt, and the program will begin running.
5. Select the "Display" tab and the Name listed under the "Device" section is the name of your graphics card.

You can also:

1. Right-click on the desktop and select "Properties"
2. In the Display Properties dialog, click on the "Settings" tab
3. Your video card manufacturer and chipset is shown below the "Display:" line
4. Look for the names NVIDIA, ATI, Intel, Matrox, SiS, S3, etc.

Updating Laptop Graphics Card Drivers

If you have a laptop, visit the laptop manufacturer's website (Dell [4], HP or Compaq [5], Toshiba [6], Sony [7], etc.) to get the most recent driver.

Updating Desktop Graphics Card Drivers

If you are using a desktop computer, visit the graphics card manufacturer's website to download the latest driver. Listed below are a few common graphics cards and links to download their drivers:

- 3DLabs [8]
- ATI [9]
- Diamond [10]
- Elsa [11]
- Intel [12]
- Matrox [13]
- nVidia [14]
- S3 [15] – Not all S3 card support OpenGL 1.5 which is required for all display options to be enabled.
- SIS [16] – Not all SIS card support OpenGL 1.5 which is required for all display options to be enabled.
- VIA [17] – Not all VIA card support OpenGL 1.5 which is required for all display options to be enabled.

Updating Windows Operating System

Many problems are resolved by keeping the windows operating system and hardware drivers up to date using the windows update site [18]. Hardware updates are often only installed if the "Custom" or "Optional" updates are included.

Updating XMS Software

Many problems are resolved by installing the latest version of XMS. Bugfixes and updates are released frequently. The updates can be downloaded at the Aquaveo Download Center [19].

Known Graphics Issues

- Issue: Graphic symbols are not displayed correctly and sometimes corrupt text lines located next to them.

  Hardware: Make: ATI Technologies Inc. Model: RADEON X600 PRO (0x5B62) Name: ATI Radeon X300/X550/X1050 Series

  Solution: Updating the driver will allow the symbols to display correctly, but the text corruption still remains.

Switch from Hardware to Software Rendering

THE FOLLOWING SHOULD BE ATTEMPTED ONLY IF THE OTHER SOLUTIONS PRESENTED DO NOT RESOLVE THE DISPLAY ISSUES

If you have updated your graphics driver and are still having problems, you can download this opengl32.dll ZIP file [20] and unzip the "OpenGL32.dll" and the "Glu32.dll" file to the directory where XMS is installed. Close and re-open XMS so this DLL is used for displaying XMS objects. Placing these DLL's in your XMS directory will fix most graphics-related issues, such as problems with displaying triangles on large TIN or DTM datasets and other problems with displaying large amounts of data. The following are known disadvantages to using this DLL for displaying:

- Displaying graphics using this DLL will likely be slower since software is used to display your graphics instead of your computer's graphics hardware. Panning, zooming, and rotating operations will be significantly slower.
- Some entities, such as symbols, are currently not displayed correctly when using this DLL. Only squares and circles will be displayed. Changing all symbol display options to squares or symbols will allow you to work around this problem. We are currently working on trying to fix this problem of symbols not displaying when using this DLL. (THIS PROBLEM HAS NOW BEEN FIXED IN SOME BETA VERSIONS OF XMS COMPILED AFTER March 31, 2009) In general, you will not want to use this DLL unless you are working with large datasets that have display issues where XMS closes unexpectedly.

Contacting Support

If you continue to experience problems after updating your graphics card drivers, contact support [1].

External Links

- Aquaveo Technical Support [1]

References

[1] http://www.logmein.com/
[2] http://www.realvnc.com/
[3] http://en.wikipedia.org/wiki/DxDiag
[4] http://www.dell.com/
[5] http://welcome.hp.com/country/us/en/support.html
[6] http://www.toshiba.com
[7] http://www.sony.com
[8] http://www.3dlabs.com/support/drivers/
[9] http://ati.amd.com/support/driver.html
[10] http://www.diamondmm.com/
[11] http://www.elsa.com/supports/download.asp
[12] http://support.intel.com/support/graphics
[13] http://www.matrox.com/mga/support/drivers/latest/home.cfm
[14] http://www.nvidia.com/content/drivers/drivers.asp
[15] http://www.s3graphics.com/drivers.jsp
[16] http://www.sis.com/support/support_prodid.htm
[17] http://www.viaarena.com/default.aspx?PageID=2
[18] http://update.microsoft.com
[19] http://www.aquaveo.com/downloads
[20] http://wms.aquaveo.com/OPENGL32.zip

Reporting Bugs

The *Help | Report A Bug* menu command can be used to send a bug report to GMS tech support. Reporting bugs helps to improve the quality of GMS. An internet connection is required to use this feature.

3. General Tools

3.1. The File Menu

The File Menu

GMS Commands
Common
File
Edit
Display
Modules
TINs
Boreholes
Solids
Mesh (2D)
Grid (2D)
Scatter Points (2D)
Interpolation (2D and 3D)
Mesh (3D)
Grid (3D)
Scatter Points (3D)
Feature Objects
GIS
Models
FEMWATER
MODAEM
MODFLOW
MODPATH
MT3D
SEAWAT
SEEP2D
T-PROGS
UTEXAS

The *File* menu is one of the standard menus and is available in all of the modules. The commands in the *File* menu are used for file input and output for the basic GMS file types, for printing, and to exit the program. The following commands are contained in the *File* menu:

Menu Command	Description
New	Deletes all data associated with all data types and all modules. It resets the status of the program to the default state that is set when the program is first launched. This command should be selected when an entirely new modeling problem is started.
Open	This command is most often used to read in project files or other GMS native files. However, this command is also used to import data or other files generated outside of GMS. Projects can either be opened into the current window which replaces the existing project, or the project can be opened into a new instance of GMS. This setting is selected in the Preferences dialog that is accessed by the ***Edit	Preferences*** command.
Import From Database	Allows users to access data stored in a database and import it.
Import From Web	Allows users to import an image from the world-wide-web.
Save	Used to save GMS projects. A project contains all of the files associated with a modeling project. When a GMS project is saved, all files associated with the data currently in memory are saved. This includes any model simulations which are open. By default the model simulation will be saved to the same location as the project. However, in the Save dialog the path for the model simulation can be specified.
Save As	Used to designate the path for saving a GMS project. It can also be used to Export data.
Edit File	Prompts for the name of a file and opens the file in a program of your choice. This command is used to edit model input files or to view output files. Output files that are part of a Solution can also be viewed by double clicking on the text file in the Project Explorer.
Page Setup	Launches the Page Setup dialog. The Page Setup dialog contains 3 tabs: Margins, Options, and Paper Size. The *Options* tab allows the user to specify the printing scale. The *Paper Size* tab allows the user to select the paper size and source. Also, the orientation, portrait or landscape, can be selected. The *Margin* tab allows the user to change the Margins. On the right side of each tab is a print preview.
Print	Printed copies of the current GMS image are generated with this command. This launches the standard Windows printing dialog.
Recent File List	Near the bottom of the file menu is a list of recently opened projects. You can have as many as 5 different files in the recent file list.
Exit	Terminates the program.

3.2. The Edit Menu

The Edit Menu

GMS Commands
Common
File
Edit
Display
Modules
TINs
Boreholes
Solids
Mesh (2D)
Grid (2D)
Scatter Points (2D)
Interpolation (2D and 3D)
Mesh (3D)
Grid (3D)
Scatter Points (3D)
Feature Objects
GIS
Models
FEMWATER
MODAEM
MODFLOW
MODPATH
MT3D
SEAWAT
SEEP2D
T-PROGS
UTEXAS

The Edit menu is one of the standard menus and is available in all of the modules. The commands in the Edit menu are used to select objects, delete objects, and set basic object and material attributes. The Edit menu contains the following commands:

Menu Command	Description
Delete	Delete the object currently selected in the Graphics Window.
Select All	Selects all items associated with the current selection tool.
Unselect All	Unselects all items associated with the current selection tool.
Invert Selection	Selects the items that were not initially selected and are associated with the current selection tool.
Select With Poly	Used to enter a polygon enclosing the items to be selected (one of the selection tools must be active). This option is useful when selecting a large irregularly shaped group of objects. To enter the polygon, click on the polygon's starting point and each intermediate point defining the polygon and double-click on the ending point. All items within the polygon will be selected.
Select With Polyline	Selects the items that intersect a polyline associated with the current selection tool.
Select From List	Some of the objects in GMS (TINs, solids, cross sections, and scatter point sets) are selected by selecting an icon that appears on the object when the selection tool for the object is active. With a large number of objects, the display of the icons can become complicated and it may be difficult to select the desired object. In such cases, this command can be used as an alternative method for selecting such objects. It brings up a list of the objects currently in memory. An object is selected by checking the box next to it in the list and then hitting the ***OK*** button.
Locate Selections	Causes a rectangle to be drawn on screen and move to surround the location of the selected items, helping to see where they are.
Data Set Calculator	The Data Calculator can be used to perform mathematical operations with data sets to create new data sets.
Properties	This command brings up the properties dialog for the currently selected item. This dialog varies depending on the selected item.
Materials	Brings up the Material Properties dialog.
Model Interfaces	Bring up a dialog containing all of the models, allowing the user to define which model menus are to be static in the menu bar.
Units	Brings Units dialog which allows the user to adjust the units for the model.
Projection...	The projection currently associated with the project can be specified via this command. Changing the projection does not alter the XYZ coordinates of the project data.
Reproject...	Reprojecting means to convert data from one coordinate system to another.
Single Point Reprojection...	This option can be used to view the coordinates of a point to be transformed.
Transform...	This command is used to scale, rotate and translate the entire project. Individual objects can be transformed by using the same command found in right-click context menus for the item in the Project Explorer.
Preferences	Brings up the Preferences dialog where the user can adjust the general preferences for GMS.
Screen Capture...	Images in the Graphics Window can be copied to the Windows clipboard by selecting the ***Copy*** command from the *Edit* menu. Once on the clipboard, the image can be copied into other applications for report generation. If the user wants to copy an image that is larger or smaller than the graphics window this can be done by editing the copy scale factor in the Preferences dialog.
Paste Text	The ***Paste*** command in the ***Edit*** window allows the user to paste tabular data directly into GMS. When this command is executed the Text Import Wizard is launched.

These were taken out of the menu

- Coordinate System || Brings up the Coordinate System dialog which allows the user to adjust the current horizontal and vertical coordinate systems.
- Coordinate Transformation || Brings up the Coordinate Transformation Wizard which allows the user to perform geographic system transformations, translations, rotations, and scaling.

Units

When building a ground water model, it is important to ensure that consistent units are used when entering model parameters. To simplify the management of model units, the user can define the units for length, time, mass, force and concentration in the **Units** dialog. A units label is placed next to each of the input fields in all the model dialogs in GMS where the units are known. For example, the units for hydraulic conductivity are length / time. If the length units are defined as "m" (meters) and the time units are defined as "d" (days) in the **Units** dialog, then the units string next to the hydraulic conductivity input field would be "m/d".

Concentration units can be defined separately, and potentially inconsistently with, mass and length units. This allows for more flexibility but can also lead to confusion so you should use care when selecting concentration units.

Unit Conversion

Generally speaking, GMS does not convert quantities from one system of units to another. GMS only displays the chosen units to help the user make sure they are being consistent. However, in a few places, GMS will use the currently defined units in it's calculations. These include:

- in the FEMWATER Fluid Properties dialog
- in the Curve Generator dialog
- when calculating the stream stage constant when saving MODFLOW, and
- in the measure tool.

Preferences

The ***Edit | Preferences*** command brings up the Preferences dialog. There are four items in the preferences list that can be selected to modify which preferences can be edited.

General

The General tab has all the of the general options in GMS.

- Restore Factory Preferences

  This button switches all settings to the factory defaults.
- Show welcome dialog on startup
- Check for newer version on startup
- Confirm Deletions

  Whenever a set of selected objects is about to be deleted, the user can choose to be prompted to confirm the deletion. This is meant to ensure that objects are not deleted accidentally.
- Show new project wizard on startup
- XMDF File Compression

  When saving XMDF files, you can use compression. The compression factor can be specified.
- Project Explorer - The Project Explorer section allows the user modify preferences related to the Project Explorer.
  - Change Module when tree selection changes.

    This option changes the current module when a item is selected in the Project Explorer. This option is on by default.

- Scroll Project Explorer when changing module.

  This option will ensure the visibility of the tree item objects in a certain module when the module is changed.
- Synchronize active data set and elevation data set for TINs, 2D Meshes, 2D Grids, and 2D Scatter Sets.

- Contouring

  Allows the user to enter a filled contour tolerance.

- North arrow paths

  Specify the path to the folder containing the North Arrows.

Model Executables

The Models page allows the user to specify the location of model executables, as well as the option to use the Model Wrapper.

- *Model Wrapper*

  GMS is a pre- and post-processor for Numerical Models. Most of these numerical models are run externally in the DOS environment. The default option is to use the Model Wrapper. The Model Wrapper "wraps" itself around the same DOS executables and gives more model feedback including graphs and tables. The Model Wrapper includes a toggle box in the bottom left that, when checked and you exit the Model Wrapper, GMS will automatically read in the results of the solution. This toggle will only be checked by default if the model converged. The option to use the Model Wrapper or the old DOS view is included in the Preferences Dialog. Certain models, including Stochastic and inverse modeling, can only be run using the Model Wrapper.

Parallel versions of MODFLOW-2000, MODFLOW-2005, and SEAWAT are shipped with GMS to run simulations. Beginning with version 8, GMS ships parallel versions where the SAMG solver has been parallelized. Even when this option is on, when running Parallel PEST with MODFLOW the serial (non-parallel) version of MODFLOW will be used since the Parallel PEST will use all of the available cores on the computer with the serial version. Also see MODFLOW preferences for an option to turn on or off the parallel version.

MODFLOW

The MODFLOW page has options related to the MODFLOW interface in GMS.

- Compress MODFLOW H5 files

  This option will force the H5 files saved with MODFLOW to be compressed. Generally this option should be turned on.

- Create h5 copy of head solution

  Turning on this option can speed up reading the MODFLOW head solution especially when there are a large number of stress periods. When this option is on, GMS writes an HDF5 copy of the MODFLOW head solution upon reading it the first time. The following times GMS reads the head solution, it doesn't take as much time.

- Default version of MODFLOW to run
  - Double precision

    When this option is on, GMS will use the double precision version of MODFLOW to run simulations. By default this option is off and GMS uses the single precision version of MODFLOW.
  - Parallel (not used by Parallel PEST)

When this option is on, GMS will use the parallel version of MODFLOW to run simulations. Beginning with version 8, GMS ships parallel versions of MODFLOW where the SAMG solver has been parallelized. Even when this option is on, Parallel PEST will use the serial (non-parallel) version of MODFLOW since the Parallel PEST will use all of the available cores on the computer.

Images / CAD

- Bitmap Scale Factor

  When the copy command is selected, a bitmap image of the screen is placed on the clipboard. The scale factor can be used to increase :or decrease the resolution of the bitmap. This scale factor also applies when saving the image as a bitmap.

- CAD Symbol Size

  The size of the CAD symbol display can be specified.

Printing

Allows the user to specify scaling Points, Lines, and Fonts while printing.

Program Mode

GMS has the option of changing the "mode" or "skin" that GMS is running. The purpose of the mode is to simplify the items available in the interface.

Available Modes

- GMS

  This is the default mode where are the tools are available to the user.

- GMS 2D

  GMS 2D is for users that are only interested in using GMS to do seepage and slope stability. In GMS 2D mode the only tools that are available are the 2D mesh and the Map. There are several changes to the GMS interface when in this mode, including:

  1. Hiding of tools and modules not in use
  2. SEEP2D model automatically initialized from conceptual model
  3. SEEP2D boundary conditions automatically mapped from conceptual model (no need to run Map -> SEEP2D)
  4. Easily create all coverage types in one step
  5. Feature object types automatically assigned based on coverage attributes
  6. Allow only one conceptual model

- GMS Site

  GMS Site is for users that are only interested in using tools for site characterization. As such the tools available in this mode are TINs, Boreholes, Solids, 2D Scatter, and Map. These tools can be used to construct solid models of stratigraphy.

Caveats

When GMS is running in a particular mode then only data associated with the available tools will be read into GMS. For example, if I have a GMS project file that contains a 3D Grid, and I am running in "GMS 2D" mode then when I attempt to read in the project file I will get a message that GMS is not enabled to read in 3D Grid data. The only data that will read into GMS in "GMS 2D" mode is data associate with the 2D Mesh and the Map.

Map

Gives the user the option to automatically perform the ***Map->SEEP2D*** command.

TINs

- Vertex options
  - Retriangulate after deleting
    - Default z-value
  - Confirm z-value
  - Interpolate for default z on interior
  - Extrapolate for default z on exterior
- Breakline options
  - Add supplementary points
  - Swap edges
- Minimum length ratio

Boreholes

- Auto-assign horizons
    - Rows per lift
    - Row overlapped between lifts
    - Sets retained between lifts
  - Measure slope from borehole tips
    - Average horizon slope
    - Horizon slope weight
    - Gap weight
- Auto-Fill cross sections
  - Prompt for vertex spacing
  - Do not redistribute vertices
  - Use default spacing
    - Default spacing/length ratio
    - Max. lense thickness ratio

Scatter sets

- 2D scatter

  Default data set value

  - Confirm date set value
- 3D scatter

  Default data set value

  Default Z value

  - Confirm date set value

2D Mesh

- Interpolate for default z on interior

  Default z (ft)
- Assign default z-value
- Prompt for z-value
- Insert nodes into triangulated mesh
- Check for coincident nodes
- Retriangulate voids when deleting

  Thin triangle aspect ratio

Graphics

- Ask which library to use if system does not support all functionality
- Autoselect the software library if system does support all functionality
- Always use system library (may not support all features)
- Always use software library (may be slower)

Edit Menu

Materials

Many of the data types supported by GMS (ex., elements, solids, borehole regions) have a material ID associated with each object. This material ID is an index into a list of material types. These material types often represent different types of soil or rock. A global list of material attributes is maintained that can be edited using the ***Materials*** command in the *Edit* menu.

The Materials are listed in a table or spreadsheet. New materials can be created by typing in the last row, or by copying and pasting from a spreadsheet, or by selecting the insert button.

Insert, delete, move up, move down buttons

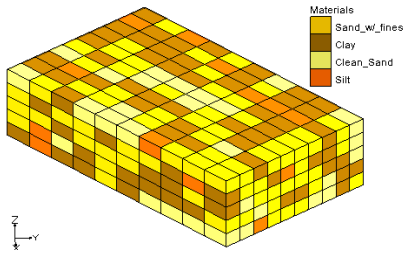
Materials have an ID, name, color and pattern. If a numerical model interface is active (MODFLOW, FEMWATER etc.), the Materials dialog will show a tab control for each active model. Additional columns in the spreadsheet are added which correspond to the appropriate properties for the model.

A legend showing all the materials can be displayed in the Graphics Window. The legend can be turned on and off in the *Display Options* dialog.

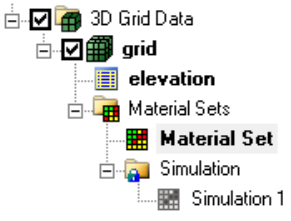
The material names, colors, patterns, and transparencies can be exported by selecting the Export button. This will save out a "material" file with a *.mat extension. This file can be imported into GMS using the File | Open command.

Material Set

A material set is similar to a data set but represents materials instead of data set values. Material sets can exist with 3D grids and meshes and can be created manually or by running a T-PROGS simulation. To create a material set manually you can right-click on the 3D grid or mesh in the Project Explorer and select the *New Material Set* command. Doing so will create a material set from the current cell or element materials. All Material sets are grouped under a Material Sets folder in the Project Explorer. Although grid cells and mesh elements (as well as other objects) have materials associated with them from the start and these material assignments can be changed on a cell-by-cell (or element-by-element) basis, a material set is not created until you specifically create it. Clicking on a material set causes the set to be applied to the grid cells (or mesh elements) and the material assignments of the cells updated.



3D grid showing materials from a material set.



A material set in the Project Explorer.

Properties

As with data sets, a material set has properties that can be viewed. Right-clicking on the material set in the Project Explorer and selecting the *Properties* command brings up the Material Set Info dialog. This dialog lists the materials in the material set, their frequency and percentage. A histogram is also drawn showing the frequency of the different materials in the set.

Viewing / editing

Clicking the Edit Materials button in the Material Set Info dialog brings up a table showing the materials in the material set, similar to viewing values in a data set. The materials can be edited and saved.

Inactive values

As with data sets, materials sets can include inactive values (starting at GMS version 8.0). Inactive materials are rendered using a material with ID -9999999.

Material set solutions

TPROGS can generate multiple material sets as part of its solution. These are stored in a material set solution folder which is locked against editing. Material set solutions can be used to run a stochastic simulation in MODFLOW.

3.2.1. Coordinate Systems

Coordinate Systems

XMS programs work in a single coordinate system. The user selects what system will be used using the *Edit | Coordinate System* menu command.

Both a horizontal and vertical system are specified. Many numerical models work in a local system, so there may not be a problem if the user does not know what coordinate system the data is referenced to. If the desired model requires a global system, or the base data is referenced to more than one system, the modeler must determine the coordinate systems involved.

Since data can be gathered and referenced to various coordinate systems, XMS allows the user to convert data from one coordinate system into another. The Coordinate Conversions page describes how to do this. The available systems and units include:

Horizontal System

The options to the right of the dialog change as the horizontal system is changed. For example, the hemisphere is required for a Geographic system and a UTM zone is required for a UTM system.

- Local
- Geographic
- UTM
- State Plane
- CPP

Local is the default horizontal and vertical system. Unless the user desires to convert data to another system, it is not necessary to change the system from *Local*.

Horizontal Units

The horizontal units can be specified for all systems except Geographic, which uses decimal degrees. **The units set here are the units used by the models such as RMA2**.

- U.S. Survey Feet
- International Feet
- Meters

Ellipsoid

The Ellipsoid options are only available for non-NAD/HPGN/CPP systems. The ellipsoid can be selected for a region of the world. Changing the ellipsoid changes the minor and major radii of the earth, measurements used in performing conversions. To set the radii for an ellipsoid not included in XMS, select the *User Defined* ellipsoid and a button appears titled *Define*. Clicking on the button brings up a dialog where the radii are defined. The Major and Minor Radii (a and b, respectively) are defined in meters only. The Minor Radius can be input directly or defined by specifying the ellipsoid flattening (1/f) or eccentricity squared (e2) variables, where

$$\frac{1}{f} = \frac{(a-b)}{a} \quad \text{and} \quad e^2 = 1 - \frac{b^2}{a^2}$$

Vertical System

The supported vertical options are for North America. If a non-NAD/HPGN horizontal system is selected, this option is disabled.

- Local
- NGVD 29
- NAVD 88

Vertical Units

The vertical unit is only used for conversions. Area and volume calculations performed by XMS use the horizontal units.

- U.S. Survey Feet
- International Feet
- Meters

External Links

- Introduction to Tidal and Geodetic Vertical Datums and Datum Computations [1] – Presentations available here [2]

References

[1] http://www.ngs.noaa.gov/corbin/class_description/Tidal_Geodetic_Datums.shtml
[2] ftp://ftp.ngs.noaa.gov/pub/corbin/datum_training/

Coordinate Conversions

Converting data from one coordinate system to another can be done using the ***Edit|Coordinate Conversions*** menu command. The command may also be accessed by right-clicking on a single entity (grid, mesh, scattered data set ...) in the Project Explorer. This allows the user to convert just that entity from one system into another (typically to the current project coordinate system).

All data will be converted from the system on the left of the dialog to the system on the right.

"Convert From" System

The "Convert From" system defines the coordinate system the data is currently referenced to. When working from the *Edit* menu, this is the system XMS is working in and is dimmed by default because it is assumed you have already specified this system. When working from the Project Explorer, this is the coordinate system of the selected entity and must be selected.

"Convert To" System

The "Convert To" system defines the system you will be working in after the conversion. When the dialog is invoked from the *Edit* menu, this is selected by the user and all data is converted from the current system to this new system. When the dialog is invoked from the Project Explorer, the "Convert To" system is dimmed because this is assumed to be the system XMS is working in and all other data is already in this system.

Restrictions

Some conversions are not allowed, such as converting between a NAD and non-NAD system. A warning is issued when conversions are not allowed.

Projections

Related Versions	
GMS	v7.0
SMS	v10.1
version note	

"Projection" refers to a map projection like UTM [1]. In XMS software, a projection can be associated with a project, and data can be reprojected from one projection to another. XMS software utilizes the Global Mapper (TM) [2] library which supports hundreds of standard projections.

Previous XMS software versions referred to projections as "coordinate systems" and reprojection as "coordinate conversion".

Project Projection

The current projection, or the projection currently associated with the project, can be specified via the *Edit|Projection* menu command. Changing the projection does not alter the XYZ coordinates of the project data.

Local Projection

Many numerical models work in local systems, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). XMS software allows for local projections that are unrelated to any standard projection.

Reproject

Reprojecting means to convert data from one coordinate system to another. For example, a 2D mesh representing the ground surface may have XYZ coordinates in a UTM system and they need to be converted to a State Plane system to be consistent with other data. Reprojecting usually results in the XYZ coordinates of the data changing, although conceptually the data is in the same place with respect to the Earth, just in a different coordinate system.

There are three basic reprojection tasks that you can do:

- Reprojecting the entire project from one system to another
- Reprojecting one object from one coordinate system to the project coordinate system
- Single point reprojection, which allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection.

Either operation brings up a dialog with two projections specified. On the left, the "Convert From" projection defines the projection the data is currently in. On the right, the "Convert To" projection defines the projection the data will be in after the operation.

Reproject everything

Reprojecting everything can be done by selecting the *Edit|Reproject* menu command. This will convert all the data loaded into the XMS application from one projection to another. In this operation, the "Convert From" projection (left side of the dialog) is dimmed by default. The user selects a new projection on the right side that all data in the application will be converted to. This operation changes the "current" projection that is being used by XMS to the newly specified "Convert To" projection.

Reproject object

This command is done on a specific geometric object (grid, mesh, scatter set, ...) by right-clicking on the entity in the Project Explorer. The object is reprojected from some projection into the current projection being used by the XMS application. In this operation, the "Convert To" projection (right side of the dialog) is dimmed by default because it represents the current projection used by the project. It is assumed that the desire is to reproject data from another projection into the current projection. The current projection is not changed in this operation.

Single Point Reprojection

Single Point Reprojection allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection. It also lets you create a feature point at the new location. This operation is accessed via the *Edit|Single Point Reprojection* menu command. It's also available in the Register Image dialog.

Restrictions

Some reprojections are not allowed, such as reprojecting between a NAD and non-NAD system. A warning is issued when the reprojection is not allowed.

Supported Projections

XMS software utilizes the Global Mapper (TM) [2] library which supports hundreds of standard projections.

References

[1] http://en.wikipedia.org/wiki/Universal_Transverse_Mercator_coordinate_system
[2] http://www.globalmapper.com/

CPP Coordinate System

A CPP (Carte Parallelo-Grammatique Projection) system is a local system. The origin of the system must be defined in latitude/longitude decimal degrees.

The conversion from of a point from latitude/longitude to CPP is:

$$\text{newpoint}_x = R * \left(\text{point}_{\text{longitude}} - \text{origin}_{\text{longitude}}\right) * \cos\left(\text{origin}_{\text{latitude}}\right)$$

$$\text{newpoint}_y = \text{point}_{\text{longitude}} * R$$

The conversion of a point from CPP to latitude/longitude is:

$$\text{newpoint}_{\text{longitude}} = \frac{\text{origin}_{\text{longitude}} + \text{point}_x}{R * \cos\left(\text{origin}_{\text{latitude}}\right)}$$

$$\text{newpoint}_{\text{latitude}} = \frac{\text{point}_y}{R}$$

R = 6378206.4 m. (Clarke 1866 major spheroid radius)

Geographic Coordinate System

A Geographic system is a latitude/longitude system defined in decimal degrees. Supported Geographic systems include:

- NAD (North American Datum) 1927 and NAD 1988
- 33 world ellipsoids and a user defined ellipsoid (i.e., Clarke 1866, WGS 1984, etc.)

The hemispheres are defined for non-NAD systems. The hemisphere cannot be changed for NAD systems (Northern, Western hemispheres).

Local Coordinate System

A local coordinate system is a system defined for a survey. Many numerical models work in local systems, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). If a portion of the data for a project is referenced to a global coordinate system, and a portion is in a local system, a transformation to convert all the data to a single system must be defined. This can be done by defining the location and orientation of the origin of the local system with reference to the global system. This allows the data to be referenced back to the global system or to another local system.

Three scenarios are possible when dealing with local coordinate systems:

1. Global to Local – When converting from a global to a local coordinate sytem, the global coordinates of the origin of the local coordinate system must be defined using the Local Origin button in the lower portion of the dialog. This dialog can also be used to enter the angle of rotation of the local coordinate axes relative to the global coordinate axes. The angle is measured ccw from the positive x-axis.
2. Local to Local – When converting from one local coordinate system to another local coordinate system, the same approach is used as when converting from a global to local coordinate sytem. The Local Origin dialog is used to define the coordinates of the origin of the new coordinate system relative to the old coordinate system.
3. Local to Global – When converting from a local to a global coordinate sytem, the Local Origin dialog is used to coordinates of the local coordinate system (which is the old system in this case) relative to the new global coordinate sytem.

Coordinate Transformation

Coordinate Transformation means changing the position (xyz coordinates) of data. There are two ways to access this feature:

- ***Edit\Coordinate Transformation*** menu command. Accessed this way the command will transform all your data.
- **Right-click on an object in the Project Explorer** and select the *Transform* command. This transforms just the object clicked on.

Coordinate Transformation Wizard

Doing either of the above will bring up the *Coordinate Transformation Wizard*. There are two steps.

Step 1 - Choose the method of transformation

The two options are:

- *Geographic system transformation*. This option is used to transform from one coordinate system to another.
- *Translate, rotate, scale*. This performs a simple transformation to move, scale, or rotate your data.

Step 2 - Choose transformation options

Geographic system transformation option

All data will be converted from the system on the left of the dialog to the system on the right.

"Convert From" System

The "Convert From" system defines the coordinate system the data is currently referenced to. When working from the Edit menu, this is the system GMS is working in and is dimmed by default because it is assumed you have already specified this system. When working from the Project Explorer, this is the coordinate system of the selected entity and must be selected. You may edit the current coordinate system by selecting the check box at the top of the dialog.

"Convert To" System

The "Convert To" system defines the system you will be working in after the conversion. When the dialog is invoked from the Edit menu, this is selected by the user and all data is converted from the current system to this new system. When the dialog is invoked from the Project Explorer, the "Convert To" system is dimmed because this is assumed to be the system GMS is working in and all other data is already in this system. In the convert to section you may select the horizontal and vertical system to be used after your conversion.

Restrictions

Some conversions are not allowed, such as converting between a NAD and non-NAD system. A warning is issued when conversions are not allowed.

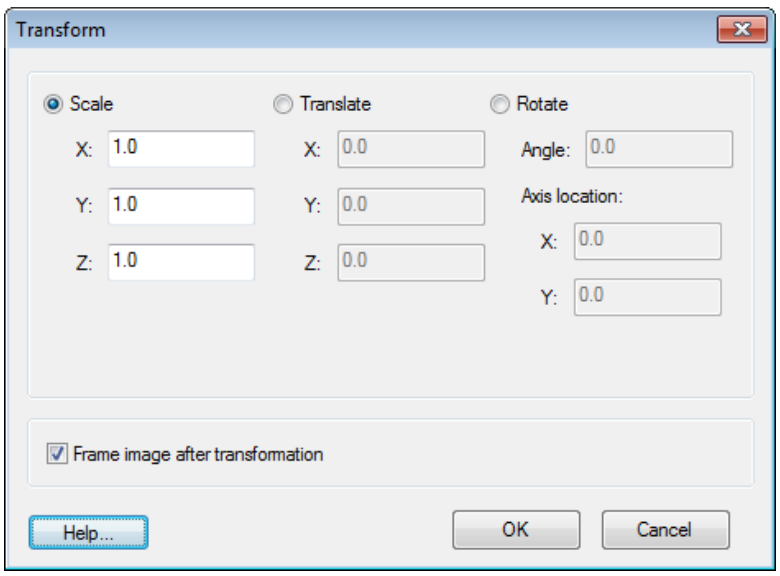
Translate, rotate, scale option

If the *Translate, rotate, scale* option was selected in step one, step two lets you chose to either translate, rotate or scale. You cannot do more than one transformation at a time - for example you cannot translate and scale at the same time. If you want to do both, you can perform the translation first, then perform the scale.

See Transform

Transform

The Transform command is used to move objects. In the Transform dialog, the transformation type can be chosen and then appropriate parameters can be entered. The following transformation types are available:



Transform Dialog

- **Scaling**: Scaling factors for the X, Y, and/or Z directions are entered. To prevent scaling a specific direction, the default value of 1.0 should be used.
- **Translation**: Translation values for the X, Y, and/or Z directions are entered. To prevent tranlation in a specific direction, the default value of 0.0 should be used.
- **Rotations**: When rotation is selected, the set of options on the right side of the dialog become available to define the center of rotation around the Z axis. The rotation will occur counter-clockwise by the specified angle around the specified center of rotation.

By default, the image will be framed after the transformation takes place. However, this can be turned off by using the Frame image after transformation option.

See Coordinate Transformation

3.3. The Display Menu

The Display Menu

GMS Commands
Common
File
Edit
Display
Modules
TINs
Boreholes
Solids
Mesh (2D)
Grid (2D)
Scatter Points (2D)
Interpolation (2D and 3D)
Mesh (3D)
Grid (3D)
Scatter Points (3D)
Feature Objects
GIS
Models
FEMWATER
MODAEM
MODFLOW
MODPATH
MT3D
SEAWAT
SEEP2D
T-PROGS
UTEXAS

The Display menu is one of the standard menus and is available in all of the modules. The commands in the Display menu are as follows:

Menu Command	Description
Display Options	Brings up the Display Options dialog
Contour Options	Opens the Contour Options dialog
Refresh	When editing the image in the Graphics Window it occasionally becomes necessary to update the display or refresh the screen by redrawing the image. Whenever possible, GMS automatically updates the display. However, in some cases small parts may be obscured by editing procedures. If so, the display can be refreshed by selecting the ***Refresh*** command from the ***Display*** menu. After selecting the ***Refresh*** command, the display process can be aborted by pressing the ESC key.
Refresh Display	When editing the image in the Graphics Window it occasionally becomes necessary to update the display or refresh the screen by redrawing the image. Whenever possible, GMS automatically updates the display. However, in some cases small parts may be obscured by editing procedures. If so, the display can be refreshed by selecting the ***Refresh Display*** command from the ***Display*** menu. After selecting the ***Refresh Display*** command, the display process can be aborted by pressing the ESC key.
Redraw Display	When zooming in and out in the Graphics Window some items such as arrows change size for increased drawing speed. This command refreshes the display and redraws these items at the size specified in the Display Options.
Frame Image	After altering the image display using the ***Zoom*** or ***Pan*** tools, the image can be centered by selecting the ***Frame Image*** command in the ***View*** menu. This command adjusts the window boundaries so that all currently visible objects just fit in the Graphics Window.
Zoom To Selections	Frames the window around whatever objects are currently selected.
Visibility	• **Hide –** A set of selected objects can be hidden by selecting the ***Hide*** command in the ***Display*** menu. Hidden objects cannot be selected. The ***Frame Image*** command only frames the image based on the visible objects. • **Show –** Hidden objects can be made visible by selecting the objects and choosing the ***Show*** command in the ***Display*** menu. This method applies to objects such as solids, TINs, and mesh layers which can be selected with an icon. Other items such as cells and elements cannot be selected when hidden but can be made visible by selecting the ***Show*** command with nothing selected. In this case, all hidden objects are made visible. • **Isolate –** In many cases it is useful to hide all of the current objects but one. One way to do this is to select all of the objects to hide (without selecting the desired object) and then select the ***Hide*** command. A quicker way to achieve the same result on one or more objects is to use the ***Isolate*** command in the ***Display*** menu. The ***Isolate*** command makes the selected items visible and automatically makes all other items of the same type invisible.
View Angle	• The objects in the Graphics Window can be rotated and viewed in three dimensions. Two angles, bearing and dip, are used to rotate the view. The bearing and dip values correspond to a rotation about the z and x axes. The bearing affects the horizontal angle (rotating the object in the xy plane), and the dip changes the vertical angle (shifting the viewing angle on the object to a higher or lower perspective). The object cannot be tilted sideways. Using only two viewing angles rather than three limits the viewing angles, but it is simpler and more intuitive. The bearing and dip angles can be explicitly defined in the View Angle dialog accessed by selecting the View Angle command from the View menu. The viewing angles can be manipulated interactively with the Rotate tool.
Window Bounds	• The region of the real world coordinate system that is mapped to the Graphics Window can be altered using the Pan and Zoom tools. It is also possible to precisely control the visible region by selecting the ***Set Window Bounds*** command from the ***View*** menu. This command brings up the **Set Window Boundaries** dialog. If the **X range to be specified (preserves aspect ratio)** option is selected, the x coordinate at the left and right and the y coordinate at the bottom of the **Graphics Window** are specified. The y coordinate at the top of the Graphics Window is not specified in order to maintain the aspect ratio. If the **Y range to be specified (preserves aspect ratio)** option is selected, the y coordinate at the top and bottom and the x coordinate at the left of the **Graphics Window** are specified. The x coordinate at the right of the **Graphics Window** is not specified in order to maintain the aspect ratio. If the **X and Y range to be specified (alters aspect ratio)** option is selected, the x coordinate at the right and left and y coordinate at the top and bottom of the **Graphics Window** are specified. Since all four coordinates are specified, the aspect ratio of the scene may be altered.

Views	• **Plan View –** Changes the viewing angle so that the user is looking down the z-axis with the x-axis horizontal and the y-axis vertical. • **Front View –** Changes the viewing angle so that the user is looking down the y-axis with the x-axis horizontal and the z-axis vertical. • **Side View –** Changes the viewing angle so that the user is looking down the x-axis with the y-axis horizontal and the z-axis vertical. • **Oblique View –** Changes the viewing angle so that the user is looking at the model at an angle of 45^{o} to each axis. • **Previous** - Restores the Graphics Window viewing parameters as they were before the last viewing command was issued (rotate, zoom, pan, etc.).
General/Ortho Mode	A command is provided in the Display menu for switching between the orthogonal and general viewing modes. The orthogonal mode is only available with 3D grids.
Convert To Cad	Converts any visible geometric data into CAD format internally within GMS. This CAD data will be saved with the project as a DWG file, or can be exported as a separate DWG or DXF file. This command is found either in the the Display Menu or by right-clicking in the empty space in the Project Explorer.
Toolbars	The Toolbars menu allows the user to hide and show the listed toolbars.
Plot Wizard	Brings up the Plot Wizard dialog.
Animate	The Animate command launches the Animation Wizard.
Play Animation	The Play Animation command launches an *.avi player that allows the user to browse for an *.avi file and play the animation.

Contour Options

The options used to generate contours can be edited by selecting the ***Contour Options*** command in the **Data** menu. The items in the Contour Options dialog are as follows:

General vs. Grid Layers

There are two tabs associated with the *Contour Options* dialog. The options described below are for the General options. The Grid Layers options are used when contouring 3D grids.

Contour Method

There are 3 main types of contours: linear, color filled, and both linear and color filled.

Linear – The default method is Normal linear contours and causes the contours to be displayed as piece-wise linear strings.

Color Fill – If the Color fill between contours option is selected, the region between adjacent contour lines is filled with a solid color. There is also a smooth option available. When this option is turned on the color between contour lines varies according to the color ramp. The transparency of the filled contours can also be adjusted.

Linear and Color Fill – This option shows linear contours on top of color filled contours.

Block Fill - This option applies only to cell centered grids. Cells are filled with a single color. The color chosen corresponds to the color ramp value associated with the value at the cell center.

Line options

Bold contours – *The Bold every...* option can be used to display contours at selected intervals with a thicker line width.

Contour labels – *The Label every...* option can be used to plot labels on contours at selected intervals. The contour label options are edited using the Contour Labels dialog.

Data Range

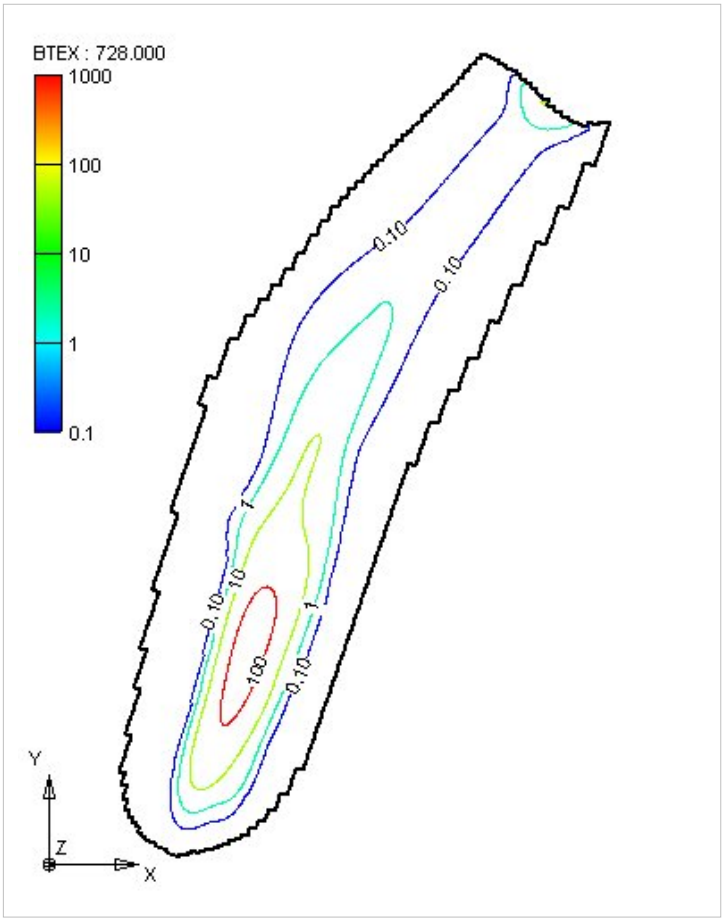
The values shown in the Data range section correspond to the maximum and minimum values in the active data set. These values are sometimes useful when choosing an appropriate contour interval.

Use each timestep's max and min – This option is only applicable is your contours are colored according to a color ramp, and your data set has more than one time step. If this option is on, the maximum and minimum color ramp colors will be displayed for every timestep. Otherwise, the maximum and minimum colors apply to the entire data set and therefore will only appear when the current data set contains the data set maximum or minimum value.

Contour Specified Range – Regardless of which option is selected for the contour interval, a maximum and a minimum contour value can be specified and the contouring can be restricted to the specified range. If desired, the fill below or fill above options can be turned on to color fill above/below the specified range.

Contour Interval

The contour interval defines the values associated with the contours. Based on the option selected, the contour values are computed and displayed in the spreadsheet. The colors of the contours will also be displayed if a Color Ramp is being used.



Four options are provided for defining contour intervals in the Contour Options dialog. The options are as follows:

Number of Contours – With the Number of Contours option, an integer is entered representing the total number of contours. The contour interval is adjusted based on the current active data set so that the contours are evenly spaced and the number of contours correspond exactly to the specified value.

Specified Interval – With the Specified Interval option, the contour interval (5, 10, 20, etc.) is entered directly by the user.

Specified Values – The Specified Values option allows the user to enter a list specific contour values. Contours are only generated at these values.

Log Interval – When contouring chemical data, it is common to have a small "hot spot" somewhere in the interior of the grid where the concentrations are several orders of magnitude higher than the rest of the grid. In such cases, the contours are all biased to the high concentration zones and no contours are drawn in the low concentration zones.

A common approach for dealing with such situations is to select a set of contour values corresponding to multiples of ten (0.01, 0.1, 1.0 10.0, etc.). This can be accomplished by explicitly assigning a set of contour intervals using the Specified Values option. The problem with this approach is that the colors assigned to the contours are still based on a linear interpolation of the data values, thus skewing the colors to the higher concentration zones.

If the Log Interval option is selected, GMS automatically assigns contour intervals as multiples of 10. Furthermore, if the color-fill or color ramp option is being used, the colors assigned to the contours are distributed in a logarithmic fashion, rather than linearly interpolated from the low to high values. A sample plot using logarithmic contour intervals is shown below:

If the Log interval option is selected, all contour intervals will be positive. No contours are drawn in areas where the data values are zero or negative. Furthermore, a "Starting value" must be entered. This value represents the minimum contour value.

Contour Labels

The ***Contour Label Options*** command in the **Data** menu is used to access the *Contour Label Options* dialog. This dialog is used to set the label color, the number of decimal places used to plot the label, and the spacing used when the labels are generated automatically. The default spacing value controls the placement of labels when labels are generated automatically.

Labels can be added to contours in one of two ways:

- 1. If the contour label option is selected in the Contour Options dialog, labels are automatically placed on the contours. The spacing of the labels is controlled with the Contour Labels dialog.
- 2. In some modules, contour labels can be added manually to contours by selecting the Contour Labels tool in the Tool Palette and clicking on the contours where labels are desired. By default, the data set value corresponding to the point that was clicked is computed and a label corresponding to the nearest contour value is drawn centered at the point that was clicked. An option can be set in the Contour Label Options dialog to use the exact value at the point that is clicked as opposed to using the nearest contour value. This option is useful to post data set value labels in regions where there are no contours.

Specified Dataset

The user specifies the dataset with the ***Select Dataset...*** button in the *Display Options|3D Grid* dialog. This brings up a new window where the specified dataset is selected. The contour options available for the specified dataset are the same as the active dataset options, yet each must be establish separately. This option is useful in comparing datasets.

Grid Layer Contour Options

When contouring 3D grid data, there are additional options available in the *Contour Options* dialog. These options let you chose the data that you want contoured. The default array used to fill cells is the active dataset. Another option for filling cells is the data colors option; this uses an input array from MODFLOW, MODPATH, MT3DMS, SEAM3D, or RT3D. Any of the input arrays from the models such as Kh, Kv, Recharge, etc. can be displayed, for any stress period. For MT3DMS/RT3D/SEAM3D, you can also specify the species.

Interpolation

GMS interpolates data from the cell centers (for cell-centered grids) to the cell corners and midsides so that the grid can be triangulated and contoured. By default, GMS uses a fast algorithm for interpolating the data that is adequate for the majority of situations. If the **Auto-interpolate contours** option is turned off, however, you can specify your own interpolation scheme. By default, GMS truncates the interpolated values to the max and min of the data, but you can turn that option off.

Animations

One of the most powerful visualization tools in GMS is animation. Animations can be created with transient or steady state data. The view angle and zoom factor can be changed during the animation as well. Animations are saved in the Windows (*.avi) format. AVI files can be played back externally to GMS using a variety of applications and can be inserted into multi-media documents and applications.

A new animation can be created by selecting the ***Animate*** command from the *Display* menu.

Animation Options

There are 4 options available when creating an animation:

Option	Type
Dataset	Transient
Particle Tracking	Transient
Cross-sections / Iso-surfaces	Steady State
Moving View	N/A

Transient vs. Steady State Animation

All of the options except Moving view are either considered Transient or Steady state. You cannot use a Transient option and a Steady state option at the same time. Moving view, however, can be used by itself or in conjunction with any combination of the other options.

The typical case for choosing to do a transient animation is when you have a transient solution generated by a model. The transient animation could illustrate how vectors, contours, fringes or iso-surfaces change at different solution times. A transient animation can also be used to show particle tracking over time, even with a steady state solution.

Steady state animation can only be performed using 3D grids or 3D meshes. A steady state animation can be used with a steady state data set, or with one time step of a transient data set. Two options are available for steady state animation: Cross-section / Iso-surface animation and Flow trace animation.

Dataset Animation

Transient dataset animation can be used with any object with a transient dataset. As each frame is generated, a set of values corresponding to the current time is loaded into memory and the image is redrawn using the current display options. Thus, if the contour display option is selected, the contours will vary from frame to frame according to the changes in the dataset.

The total number of frames generated in the animation can be defined by either matching the time steps (one frame per time step) or by using a constant interval (e.g., one frame for every two hour interval). If the Match time steps option is chosen, extra frames can be created between each time step if necessary using linear interpolation of the data values at the specified time steps.

Particle Tracking Animation Particle tracking can be animated to show how pathlines grow over time. Although it is considered a Transient option in the table above, this option can be used with a transient or a steady state solution. Only the forward tracking particle sets can be animated.

If Particle tracking is selected by itself, you can choose how many frames you want in the animation. GMS will then compute the time at which to generate each frame by dividing the total animation time evenly by the number of frames. The animation begins at the minimum start time of all forward tracking particle sets, and ends at the maximum travel time of all particles (in all forward tracking particle sets).

Cross-section / Iso-surface Animation

If the Geometric surface animation option is chosen, a cutting plane and/or an iso-surface can be animated. If the Animate cutting plane over specified XYZ range option is chosen, an x, y, and/or z cutting plane is incrementally moved through the mesh or grid from the specified beginning location to the ending location as each frame is generated. This generates an animation showing a moving cross section. If the Animate iso-surface option is chosen, a single iso-value is incrementally varied between the specified beginning and ending values and a different iso-surface is generated for each frame based on the value.

Moving View

The Moving view option can be used by itself or in conjunction with any combination of the other options. This option can be used to change how you view the data as the animation proceeds. You can define views and associate them with particular frames of the animation. When the animation is generated, GMS will interpolate between the defined views to determine what the view should be at every frame.

Flow Trace Animation

If the Animate flow trace option is chosen, a flow trace animation of the steady state vector data on 3D grid or 3D mesh cross sections is generated. Flow trace animation is a special type of animation that is similar to particle tracking. A series of particles is randomly generated on the cross sections and it is traced through time. Each particle has a limited life span. As a result, the particles appear as a series of streaks. Flow trace animation can result in highly intuitive images of a vector field.

If there is no vector dataset or cross section associated with the current 3D grid or 3D mesh, the Animate flow trace option and Flow trace options button are dimmed. The Animate flow trace option is also dimmed if the Flow trace option is not selected in the Cross Section Options dialog. If the active vector data set is transient, only the current time step is used to generate the flow trace animation.

The Avg. number of particles per triangle edit field specifies the density of particles to be generated in the animation. The Decay ratio specifies the amount of time necessary for a particle path to decay as it passes points in the cross section. The Velocity magnitude limit and Velocity direction limit specify the distance that a particle will travel between consecutive frames.

CODEC

Starting in GMS 6.5 you can choose what codec you use to create your AVI movie. GMS will search your computer for all compatible codecs and they will be available in the pull down menu.

Animation Playback

Once a new animation has been generated, GMS launches the AVI player and plays the animation. The speed of playback can be adjusted using the Speed scroll bar. The maximum speed depends on the speed of the computer and the size of the image being animated. The smaller the image, the faster the maximum playback speed.

Color Ramp

Several of the display options (contours, iso-surfaces, color-shaded vectors, etc.) use a color ramp to vary the display color based on a relative value. The color ramp can be edited by selecting the Color Ramp Options command in the Data menu. The Color Ramp Options dialog contains the following options:

Legend

If the Legend option is selected, a vertical strip of colors with a legend of corresponding data set values is displayed in the upper left corner of the Graphics Window whenever the color ramp is used to display an object (shading, color contours, etc.). The length and width of the color legend are specified with the Legend width and Legend height edit fields. The values entered for legend width and height are in screen pixels.

Palette Method

The Palette Method section lists the three basic options for defining the color ramp:

Intensity Ramp

The Intensity Ramp option defines a ramp of colors corresponding to a varying intensities of a single color.

Hue Ramp

The Hue Ramp option defines a ramp of color hues (red-yellow-green-blue-magenta) at the maximum color intensity.

User-Defined

If the User-Defined option is selected, a user-defined color palette is used to define the color ramp.

User-Defined Palettes

The user-defined color palettes are listed in the upper right corner of the Color Ramp Options dialog. A new palette is created by selecting the New button. This button brings up the New Palette dialog listing a set of options for defining the initial color palette. These colors can be edited using the Current Palette section of the Color Ramp Options dialog. An existing palette can be deleted using the Delete button.

Once a set of user-defined color palettes are created, they are saved with the project to a palette file (*.pal). The Import and Export buttons can be used to share user-defined palettes between projects.

Current Palette

The current color palette is displayed in the Current Palette section. The min and max value of the color ramp can be set by clicking and dragging the two triangles just below the color palette. For user-defined color palettes, new colors can be added, colors can be deleted, and the color associated with a color entry can be edited using the tools just below the palette.

The data value associated with a selected color can be edited either by dragging the color or by entering a new value directly. The values can be displayed as either percentages or direct values (corresponding to the active data set). The Edit Table button can be used to edit the colors and corresponding values directly in a tabular format.

Preview

The Preview section at the bottom of the Color Ramp Options dialog displays the color ramp defined by the current palette and max and min values. The Reverse button can be used to reverse the direction of the color ramp (for example, to switch from red-yellow-green-blue to blue-green-yellow-red).

3.3.1. Display Options

Display Options

Most of the data types in GMS have a set of display options that can be modified using the Display Options command in the Display menu. The Display Options command brings up the Display Options dialog. The dialog can also be brought up using the Display Options macro.

Data Type Tabs

The Display Options dialog contains a list of the data objects currently in memory. Corresponding data tabs are displayed in the dialog depending on what object type is selected in the list box. Each display feature associated with a data type is listed in the Display Options dialog. The check box next to the feature named can be toggled on or off to control whether or not the feature is to be displayed. In addition, the button next to the feature brings up a dialog that can be used to edit the display attributes of the feature (color, font, line thickness, etc.). The following table describes the display options tabs that exist for the various data types. Click on the links to learn more about the specific display options for each data type.

Image	Data Type	Tabs Also Available for Data Type
	TIN Data	
	Borehole Data	
	Solid Data	
	2D Mesh Data	SEEP2D
	2D Grid Data	
	2D Scatter Data	
	3D Mesh Data	FEMWATER, Particles, WASH123D, ADH
	3D Grid Data	MODFLOW, Particles, MT3D, UTCHEM
	3D Scatter Data	
	Map Data	MODAEM, UTEXAS
	GIS Data	
	Cross Sections	
	Lighting Options	
	Axes	
	Drawing Grid	

General Display Options

The Display Options dialog also controls several general display options in GMS. These general display options are described in the table below.

Display Option	Description
Z magnification	Occasionally an object may be very long and wide with respect to its overall depth (z dimension). In such cases, it is possible to exaggerate the z scale so that the variation in the z value is more apparent by changing the magnification factor from the default value of 1.0.
Background color	The Color dialog can be brought up to change the selected color. The selected color is displayed as the background color in the Graphics Window.
Triad options	To aide in visualization of 3D objects in oblique view, a XYZ triad can be displayed in the lower left corner of the Graphics Window. The display, size, and color of the triad can be specified in the Display Options dialog. The triad is useful in visualizing how the geometry currently defined in GMS aligns with the world coordinate system.

Drawing Grid Options

When entering new nodes or entering a polygon or polyline in plan view, it is often useful to have the coordinates snap to a uniform grid. This allows accurate placement of the objects when the desired coordinates are even multiples of some number. The properties of the drawing grid can be controlled through the Drawing Grid tab of the Display Options dialog. This dialog is accessed from the Display menu or the Display Options macro. The following table describes the display options available for the Drawing Grid.

Display Option	Description
Spacing	The Spacing edit field specifies the spacing of the grid nodes and grid lines in the drawing grid. The **Grid color** window specifies the color that is used to display the drawing grid in the Graphics Window.
Snap	If the Snap option is selected, all new vertices, nodes, points, etc., snap to the closest grid point as they are being created or when they are dragged interactively.
Display grid lines	If the Display grid lines option is selected, grid lines are displayed according to the Line spacing increment. For example, if the Grid spacing is set to 10 and the Line spacing increment is set to 5, a grid line will be drawn every 50 units. The grid line color can also be adjusted using the button next to the Display grid lines toggle.
Display grid points	If the Display grid points option is selected, grid points are displayed according to the Grid point spacing increment. The grid point color can also be adjusted using the button next to the Display grid points toggle.

Vectors

If the Vectors item in the Display Options dialog is selected for an object (TIN, Grid, or Mesh), vector plots can be generated using the active vector dataset for the object. One vector is placed at each node, cell, or vertex.

The display of vectors can be controlled using the Vector Options dialog accessed through the Vector Options command in the Data menu or from a button in the Display Options dialog. The dialog options are as follows:

Dimensions

The Auto compute length toggle is on then GMS will compute the length of the vectors. If this toggle is off then the Length can be edited. The edit fields labeled Length, Head width, Head length, and Stem width control the size and shape of the vectors. The Head width, Head length, and Stem width are expressed as a percentage of the Length of the vector.

Vary Length and/or Color

Often it is desirable to vary the display of the vectors according to the magnitude of the vector function at a current point. This can be done with the Vary length according to magnitude and Vary color according to magnitude. If the Vary length according to magnitude option is selected each vector is displayed with a length equal to the magnitude of the vector multiplied by the Scaling ratio. If the Vary color according to magnitude option is selected, the vectors with the smallest and largest magnitude are draw in the color of the lowest and highest colors on the current color ramp. All other vectors are drawn in a color which corresponds to its magnitude from the current color ramp.

Display Every Nth Vector

If the Display every _nth vector is selected, only every nth vector is drawn. This is useful when the model is so large that drawing every vector clutters the display.

Color Specified Range

It is possible to have the current color ramp vary over a specified range of the active vector data set. This is done by selecting the Color specified range option and editing the Min magnitude and Max magnitude edit fields.

2D vs. 3D Vectors

If you are in one of the orthogonal views (xy plane, xz plane, yz plane) the vectors will be displayed as 2D vectors. If you are in oblique view then the vectors are displayed a 3D vectors.

Lighting Options

This dialog allows the user to control the shading of faces in the SMS display. By default, all objects are displayed in the color specified by their attributes. However, objects such as elements, cells and triangles which cover an area, can be more intuitively understood if they are shaded as a three dimensional entity. The shading options includes two toggles, one slide bar and a light position window.

The lighting options are accessed by clicking on the *Lighting Options* item or tab in the *Display Options* dialog. The default options vary between applications, and the options may be changed, saved, and restored within the project.

Toggles

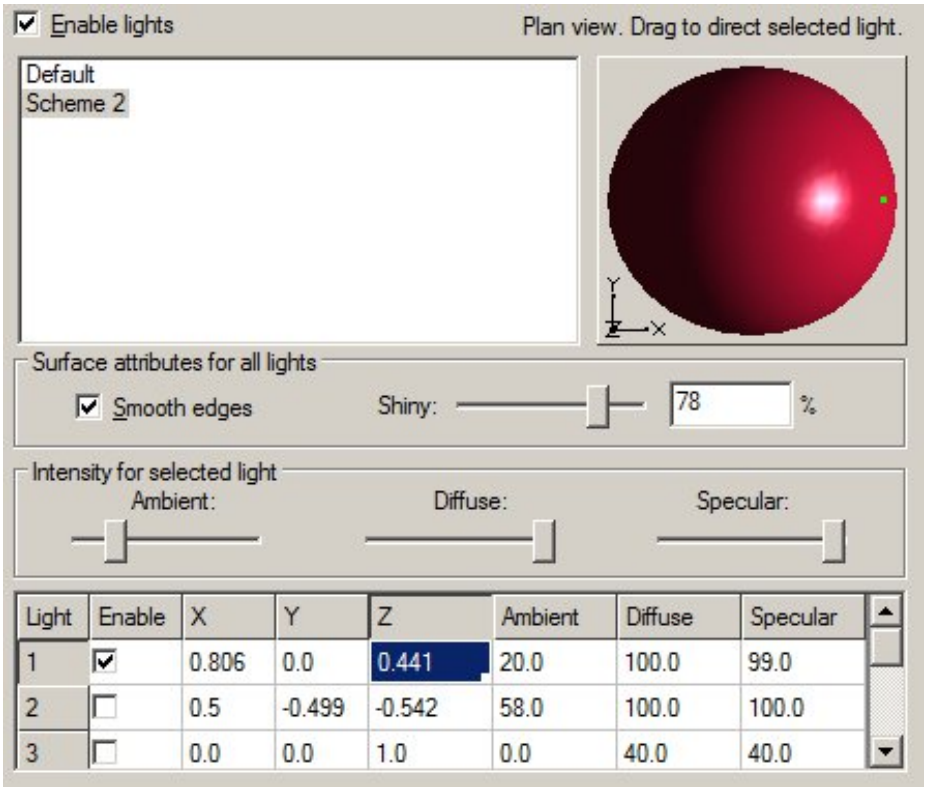
The first toggle allows the user to turn on the use of a light source. When this toggle is selected, the second toggle becomes available. The second toggle tells SMS to smooth corners between adjacent faces. This allows the faceted surface to appear as a smooth surface.

Slider

The slide bar allows the user to specify the amount of ambient light. Ambient light is the minimum intensity (brightness) to be displayed. A recommended value is between 0.2 and 0.4.

Light Position

The right side of the dialog allows the user to set the light direction and gives a preview of that direction displayed on a sphere.



The following table describes the lighting display options.

Display Option	Description
Enable lights	This check box controls whether light sources are used in the lighting process for generating lighted images. These light sources control the intensity of the colors on the lighted image and highlight the relief or geometrical variation in the surface of the objects being lighted.
Lighting list box	This list contains preset lighting schemes and highlights the scheme currently displayed.
Renaming a scheme	Double click on a scheme to begin editing its name.
Deleting a scheme	Right click on a scheme and select Delete. The final scheme cannot be deleted.
Creating a scheme	Right click on a scheme and select duplicate.
Plan view preview	This preview shows the current light scheme on a sphere in plan view, i.e., looking along the z-axis. Click or drag within the preview to direct both the diffuse and the specular components of the light currently selected in the light table. The selected light direction is shown by a dot on the sphere. A direction from in front of the sphere is shown by a green dot, and from in back by a red dot.
Smooth edges	Check this box to smooth all diffuse and specular lights of this scheme so that the surface does not appear faceted.
Shiny	Increase this value to sharpen all specular highlights of this scheme. At 100% this value turns off the specular highlight since it assumes that all specular lights are points whose reflection shrinks to a imperceptible point at maximum shinyness. At 0% this value assumes that the full intensity of the light is reflected in all directions (decrease the specular values proportionally to get a realistic effect of less and less light reflecting to the eye from each surface).
Ambient slider	Shows the Ambient value of the light currently selected in the table, and can change the value. The ambient value is light from all directions which lights each and all surfaces uniformly leaving no surface unlighted. It is most useful on surfaces facing away from directional light such as diffuse and specular light.
Diffuse slider	Shows the Diffuse value of the light currently selected in the table, and can change the value. The diffuse value is for a point light which brightens surfaces in all directions the more they face the that light, and which leaves surfaces in darkness that face away from the light.
Specular slider	Shows the Specular value of the light currently selected in the table, and can change the value. The specular value is a point light which brightens surfaces if they reflect like a mirror from the direction of the light to the direction of the viewer, and which leaves surfaces in darkness that do not have this angle of reflection.
Light table	Displays the enable, xyz position, Ambient, Diffuse, and Specular values for each of 8 lights in the current scheme, and highlights the currently selected light. Any of these values may be modified by clicking them and editing their value.
Enable column	Check these boxes to turn on each light.
X, Y, and Z columns	Edit these values or click/drag in the plan view preview sphere to change the direction of the light. These values are will be normalized to a unit direction vector.
Ambient, Diffuse, and Specular columns	Edit these values or drag their corresponding slider.

Plot Axes

The Plot Axes are a set of ruled lines oriented in either the world coordinate system or the grid coordinate system. The axes can be either 2D or 3D. The properties the plot axes can controlled through the Plot Axes tab of the Display Options dialog. The Plot Axes options are accessed by selecting the Axes item in the Display Options dialog. The Plot Axes tab is then displayed on the dialog, and is used to specify the attributes of the axes, axes ticks, axes labels, and axes numbers. The following table describes the display options available for the plot axes.

Display Option	Description
Display plot axes	If the Display plot axes toggle is selected, the plot axes will be displayed as specified and a sample of the plot axes will appear in the window in the upper left of the dialog. If the Display plot axes toggle is not selected, all plot axes will be hidden and the rest of the dialog will be dimmed.
Origin options	If the **Use world coordinate origin** option is selected, the numbers displayed on the plot axes are defined by the origin of the world coordinate system. If there is either a 2D or 3D grid currently defined in GMS, the origin of the grid can be used to define the numbering by selecting either the Use **3D grid origin** option or the **Use 2D grid origin** option. If there is not a 3D grid defined in GMS, the **Use 3D grid origin** option is dimmed. Likewise, if there is not a 2D grid defined in GMS, the Use **2D grid origin** option is dimmed. If the world coordinate origin is used, the world coordinate system will also be used to align the axes. However, if the origin of either the currently defined 2D or 3D grid is used, the plot axes can be aligned with either the world coordinate system or the local grid coordinate system. If the angle of rotation of the currently defined grid is 0.00 degrees, there is no difference between the world coordinate system and the local grid coordinate system. If there is an angle of rotation other than 0.00 degrees, the local grid coordinate system can be used by selecting the Local grid coordinates option. The world coordinate system can be used by selecting the World coordinates option.
Use exponential notation	This option controls whether or not the axes numbers are displayed using exponential notation.
Axis extents	The extents of the plot axes are specified with the controls in the spreadsheet. The **Auto scale X**, **Auto scale Y**, and **Auto scale Z** options are used to specify that the extents and spacing of the axes will be automatically calculated by GMS. If extents or spacing other than the calculated defaults are desired, any or all of the **Manual scale X**, **Manual scale Y**, and **Manual scale Z** options can be selected. If one of the Manual scale options is chosen, the min, max, tick interval, and how many ticks are to be labeled are specified for the corresponding direction. All of the directions that use the **Auto scale** option are further controlled by the **Fit to bounding volume** and the **Offset from bounding volume** options. If the **Fit to bounding volume** option is selected, the axes will be placed such that they fit tightly to the bounding volume of the currently defined objects in GMS. If the **Offset from bounding volume** option is chosen, the axes will be placed such that they are 15-20% larger than the bounding volume of the currently defined object in GMS.
Axis properties	The axis properties can be changed for each axis by right-clicking no the axis in the canvas window. The right-click menu options include **Hide axis** and **Edit Axis**. The Edit Axis options brings up a separate dialog with attributes including color and size that can be specified.

3.4. Other Tools

Annotations

Annotation Objects

The XMS application family provides a series of tools to annotate the data in an application for presentations, animations and screen shots.

These tools (annotation objects)are accessed through the Annotations Module and include:

- Images
- North Arrows
- Scale Bars
- Text
- Lines
- Ovals
- Rectangles

Screen vs World Space Layers

All annotation layers either contain objects referenced to world or screen coordinates. Objects referenced to world coordinates will change size and position on the screen with the underlying data. This is useful to identify specific locations in your model such as peir locations. Objects associated with screen coordinates do not move on the screen with the underlying data. This is useful for titles, legends such as north arrows and scale bars, and logos. Some types of annotations can only be created in screen space layers including North Arrows, Images, and scale bars.

If the first annotation object you create, could be part of a screen or world space layer SMS will ask which type of layer you wish to create and add the object to. You can create additional layers by right clicking on the *Annotation Data tree* item and selecting ***Create Screen Space Layer*** or ***Create World Space Layer***. Layers are differentiated by including an 'S' for screen space layers or 'W' for world space layers in their icons in the project explorer.

If multiple layers exist, any newly created annotation object will be placed in the "current" layer.

Annotation Object Attributes

The extents of annotation objects defined by a frame. The user defines this frame initially when creating the annotation object by left clicking at any point on the screen and dragging a rectangle with the mouse (left button still down). The display will show the frame while you drag with the mouse. (Points and lines defining degenerate frames are not allowed.) When the user creates a annotation, if the frame is too big for the window, it will be resized appropriately. Annotations can't be resized or moved even partially outside of the borders of the window. If the user resizes a annotation through a quick mouse drag and the cursor lands outside the window, the annotation will be redrawn to take up all the window space in that direction.

This frame bounds the region of the screen where the object will appear with the modeling data. The user interacts with the object by interacting with its frame and specifying its attributes or properties (see the section on selection below). The frame anchors the annotation object on the screen. This anchoring defines both the size and position of the object. The x-location, y-location, x-size and y-size are all defined independently as either a pixel value or percentage of the screen.

Horizontally, the user can position the left edge, the right edge or the center of the object. If the user positions the left edge, the object position is defined relative to the left edge of the screen. If the user positions the right edge, the object position is defined relative to the right edge of the screen. If the user positions the center of the object, the object position is defined relative to the horizontal center of the screen.

For example, the left side of the frame may be specified as 100 pixels from the left edge of the screen. Alternatively, the user may specify that the right edge of the frame should be 10% of screen width from the right edge. Finally, the user may specify that the center of the object is 100 pixels to the right of the center of the screen.

The vertical position and sizes of the object are similarly specified in the anchoring attribute of the object.

All annotation objects also have attributes. The specific attributes depend on the type of object. The attributes define color, line thickness, fill properties, associated images, etc.

Screen Space Images

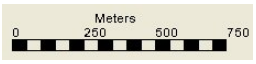
A screen space image is simply a graphics icon mapped to the screen. A typical application would be to display a company, department, or municipality logo next to the numeric model being displayed in the graphics window.

Attributes of the screen space images include:

- General anchoring attributes
- The image file being displayed as part of the project
- Whether the image is being displayed as a scaled (distorted object), scaled based on its original aspect ratio, or locked at another aspect ratio.
- Transparency – The image properties dialog have a transparency checkbox. When checked it will cause the image to be redrawn with the most used color in the image. When that it is checked, it also causes the color checkbox and the tolerance edit field to become available. If the color checkbox is checked, it will activate the color button and the color button will have the latest chosen image color painted on it or the most used color in the image, if it has not been activated before. Clicking on the down arrow part of the color button causes a color popup to be displayed with swaths of the 40 most used colors in the image or all the colors in the image, if the image has less than 40 colors. Clicking on one of those colors will cause the image to be redrawn with that color made transparent in the image. The tolerance edit field allows for variation in the matching of the red, green and blue components. The tolerance field ranges in allowable values from 0.0 to 1.0. 0.0 means the red, green and blue components must exactly match. Values higher than 0.0 indicate the degree of variation from the given color. Clicking the transparency checkbox to the off state causes the image to be redrawn with no transparency.

Scale Bars

A scale bar occupies a fixed size of the screen to display the relative size of the objects in the simulation. The user defines the minimum width of the scale bar section (in pixels), along with a minimum and maximum height of the scale (also in pixels). The XMS application adds a "Units" label (meters in the image shown below) and labels for the model distance related to the scale divisions.



The XMS applications will compute a well conditioned number to use as the scale increment that fits in the specified scale bar extents.

Attributes of the scale bar include:

- General anchoring attributes.
- The minimum spacing between distance labels.
- The minimum division width (in pixels). The XMS application determines the number of divisions based on the minimum divisiion width and the width of the frame.
- The minimum and maximum height of the scale bar object.
- The font (style, color, size) used to label the scale bar.
- Whether or not the area behind the scale bar will be filled, and if so, with what color.

North Arrows

North arrow objects consist of automatically rotating screen space images. When an XMS application is installed, at least one default North Arrow image will be included in the application's home directory. Users may create or download as many north arrow icons as desired. These icons are displayed at the specified location (anchored with the standard options), but will rotate as the view direction changes so that the "up" direction of the icon always aligns with the "North" or positive "Y" direction.

Attributes of north arrows include:

- See screen space image attributes.

Text

Text can be created in world or screen space layers.

You can enter text by clicking in the graphics window with the create text tool active.

You can set the following attributes for text:

- Font – including size
- Color
- Background – fill behind with the background color or another color
- Border and border thickness

Lines/Arrows

You can create Lines/Arrows using the Create Line Tool. Lines/Arrows can be created in screen or world space layers.

The attributes available for Lines/Arrows include:

- Type – Dashed or solid w/ thickness
- Color
- Arrowheads – location (beginning, end, both) and size

Ovals

You can create ovals by dragging a box with Create Oval tool active. Ovals can be created in world or screen space layers.

The attributes that can be specified for ovals include:

- Line type – dashed/solid with width
- Line color
- Fill and color or No Fill

Rectangles

Rectangles can be created in screen or world space layers. Their attributes are the same as those available for ovals.

Selection

The first toolbar button is used to select and set attributes for annotation objects. This requires that objects exist to be selected. In this case when you press this tool and then left click in the annotation object, the object frame will be drawn around the annotation. In addition to the frame, the XMS application displays grab handles on the corners and edges of the frame. The user modifies the rectangular shape of the annotation by dragging one of the grab handles and changes the position of the object by dragging the annotation (click at any point in the object interior).

When this tool is active the right mouse button will pop up a menu of all dialogs for that particular type of annotation. This will include at least "Attributes" which will display a dialog of the objects attributes for editing.

Viewing Annotations At Specific Time Intervals

Available in SMS 11.1 and in development for GMS, annotations can be setup to be viewed at specific time intervals. To setup annotations so they only are displayed at specified time intervals do the following:

- Right click on the Annotation layer in the tree then select *Properties...*.
- This dialog will display the *Annotation Layer Properties* dialog.
- Check the *Apply time range* checkbox
- Modify the "begin" and "end" time controls to specify the range for when annotations are visible.
- Click *Ok*.

Now film loops and time steps will only display the annotation when its within the range specified.

CAD Options

DWG and DXF files can be imported into GMS and displayed to assist in model placement or simply to enhance the display of a model.

The objects in a DWG or DXF file are organized into layers. The display of layers in a CAD drawing is controlled using the check boxes in the Project Explorer. Individual layers can be turned off/on or if you would like to turn off the display of all CAD data then uncheck the box next to the CAD folder.

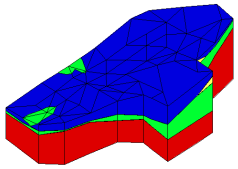
Delete Data To Delete the CAD Data right-click on the CAD data folder in the tree and select Delete from the pop-up menu. If the CAD data was imported from a file, the file is not deleted from disk.

CAD → Feature Objects CAD data can be converted to GMS feature objects by right-clicking on the CAD data folder in the Project Explorer and selecting CAD To → Feature Objects command. CAD points are turned into points, CAD lines and polylines are turned into arcs, and CAD polygons are turned into polygons. The feature objects are added to the active coverage. Once converted, the feature objects can be used to build conceptual models.

CAD → TIN A set of CAD 3D faces which have been imported to GMS can be converted to a TIN by right-clicking on the CAD data folder in the Project Explorer and selecting the CAD To → TIN command.

Convert To CAD This command converts any visible geometric data into CAD format internally within GMS. This CAD data will be saved with the project as a DWG file, or can be exported as a separate DWG or DXF file. This command is found either in the the Display Menu or by right-clicking in the empty space in the Project Explorer.

Cross Sections

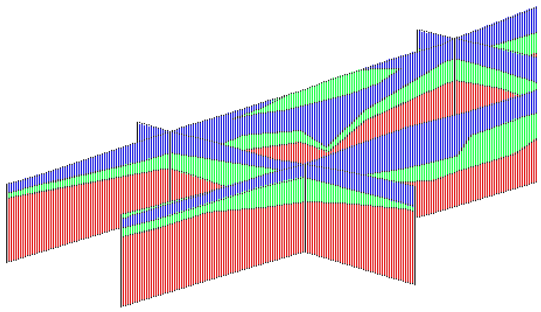
Solid Module

Solid
Solid Properties
Solid Primitives
Solid Cross Sections
More
Boundary Matching
HUF Package
Horizons to Solids
Set Operations
Solid Display Options
Solid Module Tool Palette
Solids to HUF
Solids to Layered Mesh
Solids to MODFLOW Command
Solid Commands

Cross sections—also referred to as fence diagrams—are flat surfaces used to visualize the subsurface. GMS has different types of cross section objects:

- Solid Cross Sections
- Borehole Cross Sections
- 3D Grid Cross Sections
- 3D Mesh Cross Sections

Solid Cross Sections

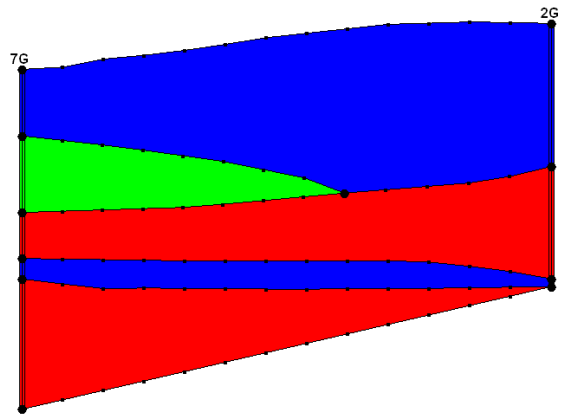
Solid cross sections can be created by "slicing" through a set of solids using the Create Cross Section tool. This can be done at any angle and the slicing can be done using a multi-segment polyline. Solid cross sections can be converted to a conceptual model. This is useful if you have a solid model of an embankment that you want to analyze using UTEXAS.



Example of fence diagrams created using solids

Borehole Cross Sections

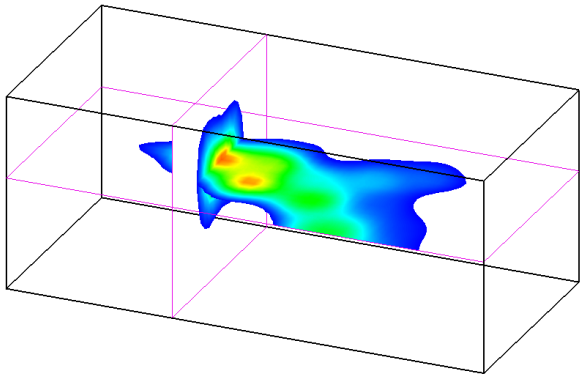
Borehole cross sections are quite different than solid, 3D grid, and 3D mesh cross sections. See the page on Borehole Cross Sections.



Borehole cross section

3D Grid Cross Sections

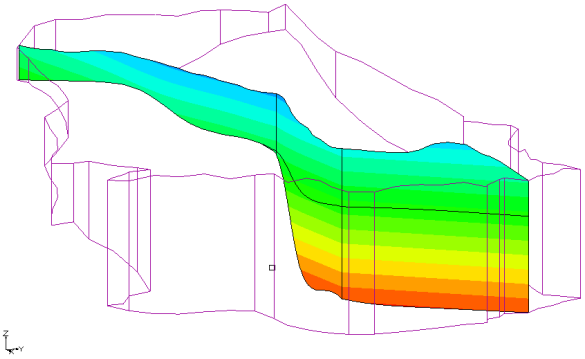
3D grid cross sections are created similar to solid cross sections. Data sets are automatically interpolated from the 3D grid to the cross sections and contours can be displayed on the cross sections.



Two cross sections cut through a 3D grid

3D Mesh Cross Sections

3D mesh cross sections are created similar to solid cross sections. Data sets are automatically interpolated from the 3D mesh to the cross sections and contours can be displayed on the cross sections.



Multi-panel cross section cut through a 3D mesh

Cross-Section Options

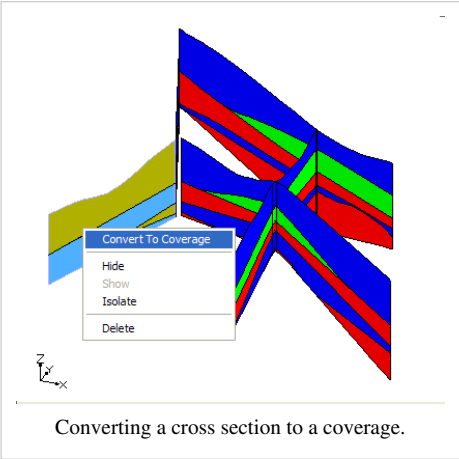
When cross sections are created from a mesh or a grid, values of the active scalar and vector data sets are interpolated to the cross sections. Whenever a new data set is chosen as the active data set for the mesh, the data values are re-interpolated to the cross sections.

The properties of all cross section data that GMS displays on the screen can be controlled through the Cross Sections tab of the Display Options dialog. This dialog is opened by right clicking on the Cross Sections entry in the Project Explorer and selecting the Display Options command. It can also be accessed from the from the Display menu or the Display Options macro. The following table describes the display options available for the cross sections.

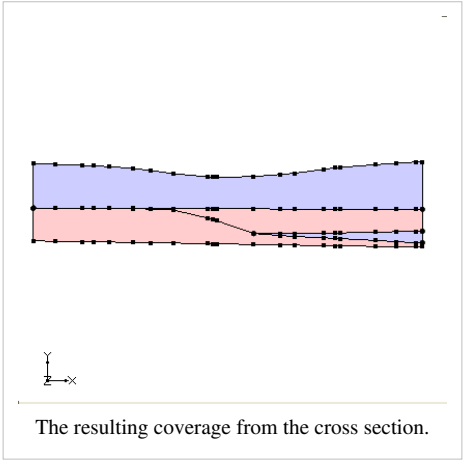
Display Option	Description
Interior edge removal	By default, the lines representing the intersection of the cross section with the faces of the cells or elements are displayed on the cross section. These lines can be hidden by selecting the Interior edge removal option.
Cross section edges	If this option is on, the lines that make up the cross section are displayed.
Cross section faces	If this option is on, then the cross section will be displayed as a set of filled polygons.
Contours	If the Contours item is selected, contours are displayed on the cross sections using the active scalar dataset.
Vectors	If the Vectors item is selected, vectors are displayed on the cross sections using the active vector dataset when the cross sections are displayed.
Flow trace	If the Flow trace item is selected, a flow trace image is texture mapped to each cross section using the active vector dataset when flow trace option is used with the Animation wizard.

Convert Cross Section to Coverage

Cross sections can be converted to coverages by right clicking on the coverage and selecting the ***Convert To Coverage*** command. The outer boundary of each material zone is converted into arcs and polygons are automatically built.



Converting a cross section to a coverage.



The resulting coverage from the cross section.

When this command is executed the user is prompted to select where the new coverage(s) should be created in the project explorer. If the user wishes to have the materials from the cross section assigned to the appropriate polygons in the coverage then the user should select an appropriate conceptual model. The conceptual model must support the assignment of materials to polygons (MODFLOW, SEEP2D/UTEXAS, FEMWATER).

Coordinate Transformation

The coordinates of points from the cross section are transformed into the XY plane. The x coordinates are calculated relative to zero where zero is defined as one end of the cross section. Currently the end of the cross section that is designated as zero is the end with the minimum Y coordinate. If the cross section is horizontal then the end of the cross section with the minimum X is used.

The x coordinates are calculated by finding the distance of the point from the end of the cross section that was set as zero. The Y coordinates are calculated by subtracting the minimum z value of the cross section from the z value of the point (y = z - z_min).

Limitations

- The cross section must be vertical.
- The cross section must be a single panel (when viewing the cross section in plan view it appears as a single line segment).

Data Sets

A dataset is a set of values associated with each node, cell, vertex, or scatter point in an object. A dataset can be steady state (one value per item) or transient (one value per item per time step). The values in the data set can be scalar values or vector values. Certain types of objects in GMS have an associated list of scalar datasets and a list of vector datasets. Each of the following objects in GMS can have both scalar a vector datasets:

- TINs
- 2D Meshes
- 2D Grids
- 3D Meshes
- 3D Grids

The following objects can only have scalar datasets:

- Borehole Sample Data
- 2D Scatter Point Sets
- 3D Scatter Point Set

The commands for manipulating datasets are located in the Data menu.

Datasets are used for both pre- and post-processing of models. For example, a scalar dataset associated with a 3D grid can represent starting values of head or values of hydraulic conductivity for a groundwater modeling problem. Another dataset associated with the same grid may represent computed head values. Datasets can be used to generate contours, iso-surfaces, vector plots, and animation sequences.

Generating Datasets

Datasets can be generated in a variety of ways. They can represent output from a groundwater model (head, drawdown, etc.). They can represent tabular values in a text file entered by the user or exported from another application such as a GIS. They can be created by interpolating from a scatter point set to a TIN, grid, or mesh. Datasets can also be generated by performing mathematical operations on other datasets with the Data Calculator.

One advantage of the dataset list approach for managing information is that it facilitates transfer of information between different types of models or models with differing resolution. This is accomplished through scatter point sets and interpolation. TINs, borehole contacts, borehole sample data, grids, and meshes can all be converted to a 2D or 3D scatter point set. When an object is converted to a scatter point set, all scalar datasets associated with the object are copied to the new scatter point set. The datasets can then be transferred from the scatter point set group to other objects of any type using one of the supported interpolation schemes.

Importing/Exporting Datasets A dataset can be exported by right-clicking on the dataset in the Project Explorer and select the Export command. The Save As dialog has three filter types that can be exported:

- Binary Dataset Files (*.dat)
- ASCII Dataset Files (*.dat)
- HDF5 Dataset Files (*.h5)

A dataset can be imported by right-clicking on any of the objects explained above in the Project Explorer and select the Import Dataset command. This command brings up the File Open dialog.

Project Explorer

Datasets are displayed and managed in the Project Explorer. Click here to learn about datasets in the Project Explorer.

Active Dataset

Each object (TIN, Grid, Mesh, or Scatter Point Set) in GMS has a set of values which is designated as the "active data set." The active dataset is an important part of model visualization in GMS. Each time the display is refreshed, the contours and other display features are generated using the active dataset.

The active dataset and time step are displayed in the Project Explorer. Left-clicking on a solution or dataset in the Project Explorer makes that item "active" and the display is automatically updated in the GMS window. The table below shows the icons displayed next to the active datasets.

active scalar dataset
active vector dataset
active ccf dataset

Dataset Info

Selecting the ***Properties*** command in right click menu on a dataset in the Project Explorer will bring up the Dataset Info Dialog. The components of the dialog are as follows:

Statistics

A set of statistics related to the active dataset is displayed on the left side of the Dataset Info dialog. the statixtics for all time steps can be shown by selecting the *Show all time step stats*. toggle. A histogram of the data values is displayed on the right side. Right-clicking on the plot and selecting the ***Plot Data*** command brings up the X-Axis Precision dialog. This dialog enables users to set the precision of the x-axis of the Histogram.

View/Edit Values

For datasets associated with model solutions, the data values can be displayed in a spreadsheet using the View Values button. For all other datasets, the button label changes to Edit Values and the spreadsheet can be used to change the dataset values.

Active/Inactive Flags

If the currently selected module is the 3D Grid module, the active/inactive status of cells can be changed by selecting the Edit button at the bottom left of the dialog. This brings up the *Active/Inactive Flags* dialog.

Date/Time Display

For a transient data set, the time values can be displayed in the Time Step Window using either a relative time format (e.g., 100.0) or using a date/time format (e.g., 1/12/1998 3:23:48). The relative times are computed using a reference time that is defined for the model (MODFLOW, FEMWATER etc.). The reference time represents the date/time corresponding to t=0.

Datasets in the Project Explorer

The solutions and data sets associated with a particular object (grid, mesh, scatter point set, etc.) are managed using the Project Explorer.

Datasets

	scalar dataset	**active scalar dataset**
	vector dataset	**active vector dataset**
	ccf dataset	**active ccf dataset**

3 different types of datasets are shown in the Data Tree: scalar, vector, and cff. The icons used to identify the datasets are shown below.

If the active dataset is transient then the time steps are displayed in the Time Step Window.

Solutions

Solutions are output from a numerical model that GMS supports. Solutions are shown in the Project Explorer as a folder with an lock on it . If a solution is transient then the time steps are displayed in the Times Step Window. Datasets can not be moved into or out of a solution folder. Also, the solution may contain text files such as the *.out and *.glo files produced by MODFLOW. These files can viewed by right clicking on the item and selecting **View File** from the pop up menu, or you can double click on the item.

Clicking on a solution or dataset makes that item "active" and the display is automatically updated in the GMS window.

Folder

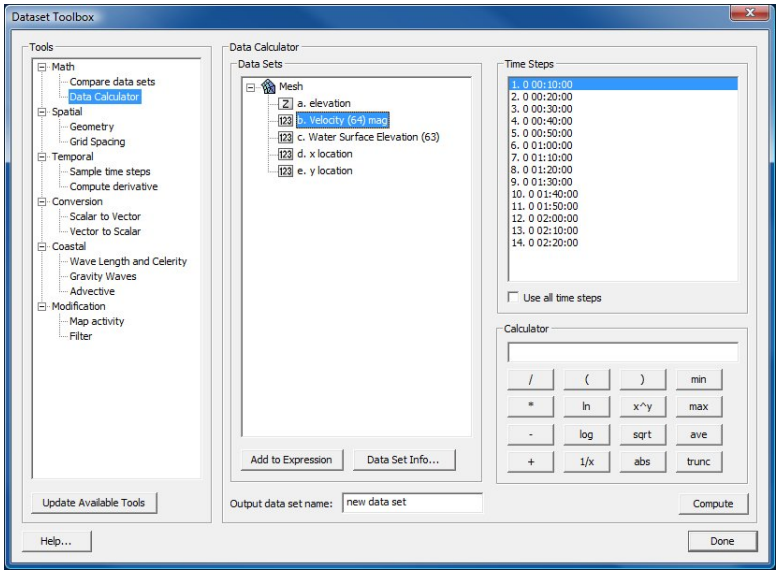
The data ets and solutions are organized by folders. The user can create new folders and move datasets, solutions, and folders to other folders anywhere on the Project Explorer. Folders can be created by right clicking on the certain items in the Project Explorer and selecting **New Folder** in the menu. A dataset or folder can be deleted simply by selecting the folder and selecting the Delete key or by right-clicking on the item and selecting the **Delete** option in the corresponding pop-up menu. Statistics about a dataset can be viewed by right-clicking on a dataset and selecting the Properties option from the pop-up menu.

Data Calculator

At a glance

- Performs mathematical calculations on scalar datasets
- Calculations can include any number of scalar datasets and user supplied numbers
- Useful for computing derived values such as Froude numbers
- Useful for comparing scalar datasets

The Data Calculator can be used to perform mathematical operations with datasets to create new datasets. The Data Calculator is accessed by selecting the ***Data Calculator*** command from the *Data* menu. The components of the Data Calculator are as follows:



Data Calculator

Expression Field

The most important part of the Data Calculator is the Expression field. This is where the mathematical expression is entered. The expression should be formulated using the same rules that are used in formulating equations in a spreadsheet. Parentheses should be used to clearly indicate the preferred order of evaluation. There is no limit on the length of the expression. The operators in the expression should be limited to the operators shown in the middle of the Data Calculator. The operands in the expression should consist of user-defined constants (e.g., 3.14159), or datasets.

List of Datasets

All of the datasets associated with the active object (TIN, Grid, Mesh, or Scatter Point Set) are listed at the top of the Data Calculator. If a transient dataset is highlighted, the time steps are listed on the right side of the Data Calculator. When a dataset is used in an expression, the name of the dataset should NOT be used. Rather, the letter associated with the dataset should be used. For example, if a dataset is listed as "b. head1", the dataset is referenced in the expression simply as "b"

When a transient dataset is used in an expression, either a single time step or the entire sequence of time steps may be used. For example, the expression "abs(d:100)" creates a single (steady state) dataset representing the absolute value of the dataset at time = 100.0. However, the expression "abs(d:all)" creates a transient dataset representing the absolute value of each of the time steps in the original dataset.

Result Name

When an expression is evaluated, a new dataset is created and the name of the new dataset is designated in the *Result* field.

Operators

The allowable operators are listed in the middle of the dialog. Selecting one of the operator buttons adds the selected operator to the end of the expression. However, the operators can also be typed directly in the expression field. The function of each of the operators is as follows:

Operator	Function
" + "	Add
" - "	Subtract
" * "	Multiply
" / "	Divide
" ("	Left Parenthesis
") "	Right /Parenthesis
" log(x) "	The base 10 logarithm of a dataset
" ln(x) "	The natural logarithm of a dataset
" x^a "	(x) raised to the (a) power. (x) and (a) can be any mixture of constants and datasets
" abs(x) "	The absolute value of a dataset
" sqrt(x) "	The square root of a dataset
" ave(x,y) "	The average of two datasets
" min(x,y) "	The minimum of two datasets
" max(x,y) "	The maximum of two datasets
" trunc(x,a,b) "	Truncates a dataset (x) so that all values are >= a and <= b
" 1/(x) "	The inverse of (x) - Only available in SMS

Operating With Transient Datasets

Each argument in the operators listed in the table above may be:

- A steady state (1 time step) dataset
- A specified time step of a transient dataset (i.e., x:#). In this case the # represents the index of the time step as specified in the time step window.
- A transient time step (i.e., x:all). These operations are only valid if all arguments have matching time step values. In this case, the result will be a new transient dataset with identical time values as the arguments.

The data calculator supports an alternate format for computing attributes of a transient dataset. This alternate format applies to three of the operators. These operators compute a single time step (steady state) dataset representing the spatially varied attribute operating on all the time steps.

Operator	Function
" ave(x:all) "	The average at each location of all time steps in the dataset
" min(x:all) "	The minimum at each location of all time steps in the dataset
" max(x:all) "	The maximum at each location of all time steps in the dataset

Compute Button

Once an expression is formulated and a name for the resulting dataset has been specified, the expression can be evaluated by selecting the **Compute** button. At this point, the dataset is created and the name of the new dataset should appear in the list of datasets.

Related Links

- Datasets (GMS)
- Datasets (SMS)
- Datasets (WMS)
- Dataset Toolbox (SMS)

Display Theme

A Display Theme is a collection of display options. The user can create a display theme by right-clicking in the blank space of the Project Explorer and selecting the *New | Display Theme* menu command. This will bring up the Display Theme Properties dialog. In this dialog the user can name the display theme, select which display options to include in the display theme, and enter the names of items that will automatically use the display theme. For more information on display themes see the Display Themes tutorial here.

XY Series Editor

The XY Series Editor is a special dialog that is used to generate and edit curves defined by a list of x and y coordinates. The curve can be created and edited by directly editing the xy coordinates using a spreadsheet list of the coordinates. An entire list of curves can be generated and edited with the Editor and curves can be imported from and exported to text files for future use. You can also paste your xy data directly to the spreadsheet.

The XY Series Editor is used in GMS, SMS, and WMS. It was designed to be general in nature so that it could be used anywhere that a curve or function needs to be defined. In some cases, the x values of the curve must correspond to a pre-defined set of values. For example, the x values may correspond to a set of time steps whose interval is established in a separate dialog. In such cases, the x fields cannot be edited but the y values associated with the pre-defined x values can be edited. In other cases, there is no limit on the number of x values or on the x spacing and both the x and y values can be edited.

The XY Edit Fields

The two vertical columns of edit fields on the left side of the dialog are for direct editing of the xy series values. A pair of application specific titles appears at the top of the columns.

The buttons below the xy edit fields are used to manipulate the values in the edit fields. The buttons are as follows:

Use dates/times For selected situations such as entering time series data in the Map module, it is useful to enter the data in date/time format. Checking this toggle allows the x values in your curve to be entered in date/time format.

Import/Export Buttons The Import and Export buttons allow the user to read in or save an xy series file.

The XY Series Plot

The window in the upper right hand corner of the XY Series Editor is used to plot the curve corresponding to the xy values in the edit fields. As each value in the edit fields is edited, the corresponding point on the curve is adjusted instantaneously. Plot options are accessed by right-clicking on the plot.

Related Pages

- GMS main page
- SMS main page
- WMS main page

4. Interpolation

4.1. Introduction

Interpolation

GMS contains a powerful suite of interpolation tools. You can interpolate to TINs, 2D meshes, 2D grids, 3D meshes, and 3D grids. The following types of interpolation are available in GMS:

- Linear
- Inverse Distance Weighted
- Clough-Tocher
- Natural Neighbor
- Kriging
- Gaussian Field Generator

GMS also supports Jackknifing, which is used to compare interpolation schemes.

In addition to interpolating scalar values, GMS also supports interpolation of materials with T-PROGS. The T-PROGS software is used to perform transition probability geostatistics on borehole data.

How To Interpolate in GMS Interpolation is performed using the 2D Scatter Points and the 3D Scatter Points. To interpolate values from a scatter set you either right-click on a scatter set in the Project Explorer and select the ***Interpolate to*** command or select the command from the Interpolation menu. The commands in the **Interpolation** menu act on the "active" item in the Project Explorer.

Interpolation Commands

Once a 3D interpolation scheme has been selected and the appropriate parameters for the selected scheme have been input, the data set of the active scatter point set can be interpolated to another object. During the interpolation process, a new data set is constructed for the target object containing the interpolated values. A separate interpolation command is provided for interpolating to each of the target objects. The interpolation commands are found in the Interpolation menu. The commands are as follows:

- **Interpolate → Active TIN**

  The to Active TIN command interpolates to the vertices of the active TIN.

- **Interpolate → 2D Mesh**

  The to 2D Mesh command interpolates to the nodes of the 2D finite element mesh.

- **Interpolate → 2D Grid**

  The to 2D Grid command interpolates to the 2D finite difference grid. The interpolation is done either to the grid nodes or to the grid cell centers depending on whether the grid is a mesh-centered or cell-centered grid. (See 2D Grid Types)

- **Interpolate → 3D Mesh**

  The to 3D Mesh command interpolates to the nodes of the 3D finite element mesh.

- **Interpolate → 3D Grid**

  The to 3D Grid command interpolates to the 3D finite difference grid. The interpolation is done either to the grid nodes or to the grid cell centers depending on whether the grid is a mesh-centered or cell-centered grid. (See 2D Grid Types)

- **Interpolate → MODFLOW Layers**

  The Interpolate to MODFLOW Layers commmand lets you interpolate from 2D scatter data to MODFLOW data: top and bottom layer elevations, LPF array data (HK, VK, etc), recharge.

- **Gaussian Simulation Options**

  Gaussian Sequential Simulation (GSS) is a form of Kriging that can only be used for 2D interpolation and only works when interpolating to 3D cell-centered grids.

- **Jackknifing**

  3D Jacknifing identical to 2D Jacknifing.

When one of the interpolation commands is selected, the Interpolate dialog appears.

3D Interpolation Options

3D scatter point sets are used for interpolation to other data types such as grids and meshes. Interpolation is useful for such tasks as iso-surface rendering or setting up input data for a model. Since no interpolation scheme is superior in all cases, several interpolation techniques are provided in GMS.

The basic approach to performing an interpolation is to select an appropriate interpolation scheme and interpolation parameters, and then interpolate to the desired object using one of the interpolation commands (to 3D Grid, to 3D Mesh, etc.) described below.

The interpolation options are selected using the 3D Interpolation Options dialog which is accessed through the ***Interp. Options*** command in the *Interpolation* menu. Once a set of options is selected, those options are used for all subsequent interpolation commands. The options in the 3D Interpolation Options dialog are as follows:

Active Dataset

Interpolation is always performed using the active dataset of the active scatter point set. The active dataset is normally selected in the Project Explorer. The name of the current active dataset is listed at the top of the 3D Interpolation Options dialog.

If the active dataset is transient then more interpolation options are available. (see Steady State vs. Transient Interpolation)

Interpolation Methods

The following methods are supported for 3D interpolation in GMS:

- Inverse Distance Weighted Interpolation
- Kriging

Log interpolation is also supported.

Anisotropy

Sometimes the data associated with a scatter point set will have directional tendencies. The azimuth and horizontal anisotropy allow the user to take into account these tendencies.

Vertical Anisotropy

In 3D, vertical anisotropy is also available. In previous versions of GMS the user could enter a Z scale. Vertical anisotropy is 1 over the Z scale. This notation was changed to be consistent with Kriging.

Occasionally, scatter point sets are sampled along vertical traces. In such cases, the distances between scatter points along the vertical traces are an order of magnitude smaller than the distances between scatter points along the horizontal plane. For example, if the scatter point set was obtained from borehole data, the distance between scatter points may be a few centimeters, whereas the distance between boreholes may be several meters. This disparity in scaling causes clustering and can be a source of poor results in some interpolation methods.

The effects of clustering along vertical traces can be minimized using the vertical anisotropy option in the 3D Interpolation Options dialog. The Z coordinate of each of the scatter points is multiplied by 1 / the vertical anisotropy parameter prior to interpolation. Thus, if the vertical anisotropy parameter is less than 1.0, scatter points along the same vertical axis appear farther apart than they really are and scatter points in the same horizontal plane appear closer than they really are. As a result, points in the same horizontal plane are given a higher relative weight than points along the Z axis. This can result in improved accuracy, especially in cases where the horizontal correlation between scatter points is expected to be greater than the vertical correlation (which is typically the case in

soils since soils are deposited in horizontal layers).

Assign default extrapolation value to hidden objects

This option will assign the default extrapolation value to all cells that are hidden using the ***Hide*** command in the *Display* | *Visibility* menu option (see The Display Menu).

Truncation

When interpolating a set of values, it is sometimes useful to limit the interpolated values to lie between a minimum and maximum value. For example, when interpolating contaminant concentrations, a negative value of concentration is meaningless. However, many interpolation schemes will produce negative values even if all of the scatter points have positive values. This occurs in areas where the trend in the data is toward a zero value. The interpolation may extend the trend beyond a zero value into the negative range. In such cases it is useful to limit the minimum interpolated value to zero. Interpolated values can be limited to a given range by selecting the Truncate values option in the 3D Interpolation Options dialog.

Steady State vs. Transient Interpolation

If the active dataset happens to be a transient dataset, two options are available:

1. Steady state interpolation can be performed using only the selected time step of the active dataset.
2. Transient interpolation can be performed using all of the time steps.

By default, only the selected time step is used. The time step is shown next to the data set name at the top of the dialog. All of the time steps can be selected by selecting the Use all time steps toggle next to the Time step combo box. If all time steps are chosen, GMS begins with the first time step in the list and repeatedly interpolates from the scatter point set to the target object, one time step at a time, for all of the time steps. As a result, a dataset is created on the target object with a set of time steps matching the time steps on the scatter point set.

When performing transient interpolation with the kriging option, special care should be taken with regard to the variogram. Since each time step represents a separate set of data, technically, a separate variogram (or set of variograms) should be created for each time step (GMS stores a separate variogram for each step). This can be accomplished by selecting each time step one at a time using the Time step combo box at the top of the Interpolation Options dialog, and creating a new variogram for each time step.

4.2. Linear

Linear

If the linear interpolation scheme is selected, the 2D scatter points are first triangulated to form a temporary TIN. The TIN is a network of triangles connecting the scatter points together. It is used to interpolate from the scatter points to another object such as a grid or a mesh.

The equation of the plane defined by the three vertices of a triangle is as follows:

$$Ax + By + Cz + D = 0$$

where A, B, C, and D are computed from the coordinates of the three vertices (x1,y1,z1), (x2,y2,z2), & (x3,y3,z3):

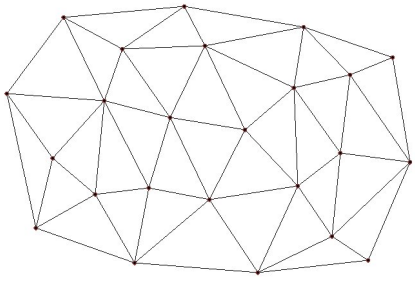
$$A = y_1(z_2 - z_3) + y_2(z_3 - z_1) + y_3(z_1 - z_2)$$
$$B = z_1(x_2 - x_3) + z_2(x_3 - x_1) + z_3(x_1 - x_2)$$
$$C = x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)$$
$$D = -Ax_1 - By_1 - Dz_1$$

The plane equation can also be written as:

$$z = f(x,y) = -\frac{A}{C}x - \frac{B}{C}y - \frac{D}{C}$$

which is the form of the plane equation used to compute the elevation at any point on the triangle.

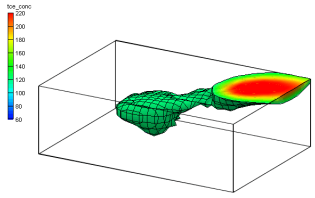
Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the linear interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the default extrapolation value entered at the bottom of the Interpolation Options dialog. The figure below shows a 2D scatter point set (small red triangles in GMS) being interpolated to a 2D grid. The green lines represent a TIN constructed from a scatter point set. The thick blue line represents the convex hull of the dataset. No extrapolation will occur outside of this thick blue line (picture is not representative of this description).



Convex Hull of a Scatter Point Set

4.3. Inverse Distance Weighted

Inverse Distance Weighted

3D Scatter Point Module

3D Scatter Point
Interpolating with 3D Scatter Points
Converting 3D Scatter Points to Other Data Types
Bounding Grid
More
3D Scatter Point Display Options
3D Scatter Point Tool Palette
Active/Inactive Points
Inverse Distance Weighted
3D Scatter Point Commands

One of the most commonly used techniques for interpolation of scatter points is inverse distance weighted (IDW) interpolation. Inverse distance weighted methods are based on the assumption that the interpolating surface should be influenced most by the nearby points and less by the more distant points. The interpolating surface is a weighted average of the scatter points and the weight assigned to each scatter point diminishes as the distance from the interpolation point to the scatter point increases. Several options are available for inverse distance weighted interpolation. The options are selected using the Inverse Distance Weighted Interpolation Options dialog. This dialog is accessed through the Options button next to the Inverse distance weighted item in the 2D Interpolation Options (3D Interpolation Options) dialog. The options in the dialog are as follows:

- Shepards Method
- Gradient Plane Nodal Functions
- Quadratic Nodal Functions
- Subset Definition

Shepards Method

The simplest form of inverse distance weighted interpolation is sometimes called "Shepard's method" (Shepard 1968). The equation used is as follows:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i$$

where n is the number of scatter points in the set, f_i are the prescribed function values at the scatter points (e.g., the data set values), and w_i are the weight functions assigned to each scatter point. The classical form of the weight function is:

$$w_i = \frac{h_i^{-p}}{\sum_{j=1}^{n} h_j^{-p}}$$

where p is an arbitrary positive real number called the weighting exponent and is defaulted to 2. The weighting exponent can be modified by turning on the Use classic weight function option. h_i is the distance from the scatter point to the interpolation point or

$$h_i = \sqrt{(x - x_i)^2 + (y - y_i)^2}$$

where (x,y) are the coordinates of the interpolation point and (x_i,y_i) are the coordinates of each scatter point. The weight function varies from a value of unity at the scatter point to a value approaching zero as the distance from the scatter point increases. The weight functions are normalized so that the weights sum to unity.

Although the weight function shown above is the classical form of the weight function in inverse distance weighted interpolation, the following equation is used in GMS:

$$w_i = \frac{\left[\frac{R-h_i}{Rh_i}\right]^2}{\sum_{j=1}^{n}\left[\frac{R-h_i}{Rh_i}\right]^2}$$

where h_i is the distance from the interpolation point to scatter point i, R is the distance from the interpolation point to the most distant scatter point, and n is the total number of scatter points. This equation has been found to give superior results to the classical equation (Franke & Nielson, 1980).

The weight function is a function of Euclidean distance and is radially symmetric about each scatter point. As a result, the interpolating surface is somewhat symmetric about each point and tends toward the mean value of the scatter points between the scatter points. Shepard's method has been used extensively because of its simplicity.

3D Interpolation

The 3D equations for Shepard's method are identical to the 2D equations except that the distances are computed using:

$$h_i = \sqrt{(x - x_i)^2 + (y - y_i)^2 + (z - z_i)^2}$$

where *(x,y,z)* are the coordinates of the interpolation point and (x_i,y_i,z_i) are the coordinates of each scatter point.

Gradient Plane Nodal Functions

A limitation of Shepard's method is that the interpolating surface is a simple weighted average of the data values of the scatter points and is constrained to lie between the extreme values in the data set. In other words, the surface does not infer local maxima or minima implicit in the data set. This problem can be overcome by generalizing the basic form of the equation for Shepard's method in the following manner:

$$F(x,y) = \sum_{i=1}^{n} w_i Q_i(x,y)$$

where Q_i are nodal functions or individual functions defined at each scatter point (Franke 1982; Watson & Philip 1985). The value of an interpolation point is calculated as the weighted average of the values of the nodal functions at that point. The standard form of Shepard's method can be thought of as a special case where horizontal planes (constants) are used for the nodal functions. The nodal functions can be sloping planes that pass through the scatter point. The equation for the plane is as follows:

$$Q_i(x,y) = f_x(x - x_i) + f_y(y - y_i) + f_i$$

where f_x and f_y are partial derivatives at the scatter point that have been previously estimated based on the geometry of the surrounding scatter points. Gradients are estimated in GMS by first triangulating the scatter points and computing the gradient at each scatter point as the average of the gradients of each of the triangles attached to the scatter point.

The planes represented by the above equation are sometimes called "gradient planes". By averaging planes rather than constant values at each scatter point, the resulting surface infers extremities and is asymptotic to the gradient plane at the scatter point rather than forming a flat plateau at the scatter point.

3D Interpolation The 3D equivalent of a gradient plane is a "gradient hyperplane." The equation of a gradient hyperplane is as follows:

$$Q_i(x,y,z) = f_x(x - x_i) + f_y(y - y_i) + f_z(z - z_i) + f_i$$

where f_x, f_y, and f_z are partial derivatives at the scatter point that are estimated based on the geometry of the surrounding scatter points. The gradients are found using a regression analysis which constrains the hyperplane to the scatter point and approximates the nearby scatter points in a least squares sense. At least five non-coplanar scatter points must be used.

Quadratic Nodal Functions

The nodal functions used in inverse distance weighted interpolation can be higher degree polynomial functions constrained to pass through the scatter point and approximate the nearby points in a least squares manner. Quadratic polynomials have been found to work well in many cases (Franke & Nielson 1980; Franke 1982). The resulting surface reproduces local variations implicit in the dataset, is smooth, and approximates the quadratic nodal functions near the scatter points. The equation used for the quadratic nodal function centered at point k is as follows:

$$Q_k(x,y) = a_{k1} + a_{k2}(x - x_k) + a_{k3}(y - y_k) + a_{k4}(x - x_k)^2 + a_{k5}(x - x_k)(y - y_k) + a_{k6}(y - y_k)^2$$

To define the function, the six coefficients $a_{k1}..a_{k6}$ must be found. Since the function is centered at the point k and passes through point k, we know beforehand that $a_{k1}=f_k$ where f_k is the function value at point k. The equation simplifies to:

$$Q_k(x,y) = f_k + a_{k2}(x - x_k) + a_{k3}(y - y_k) + a_{k4}(x - x_k)^2 + a_{k5}(x - x_k)(y - y_k) + a_{k6}(y - y_k)^2$$

Now there are only five unknown coefficients. The coefficients are found by fitting the quadratic to the nearest NQ scatter points using a weighted least squares approach. In order for the matrix equation used to solve for the coefficients to be stable, there should be at least five scatter points in the set.

3D Interpolation

For 3D interpolation, the following equation is added to the quadratic nodal function:

$$+ a_{k5}(x - x_k)(y - y_k) + a_{k6}(x - x_k)(z - z_k) + a_{k7}(y - y_k)(z - z_k)$$

To define the function, the ten coefficients $a_{k1}..a_{k10}$ must be found. Since the function is centered on point k, we know that $a_{k1}=f_k$ where f_k is the data value at point k. The equation simplifies to:

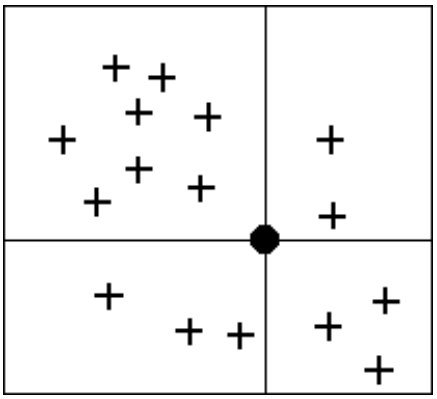
$$Q_k(x,y) = f_k + a_{k2}(x - x_k) + a_{k3}(y - y_k) + a_{k4}(z - z_k)$$

Now there are only nine unknown coefficients. The coefficients are found by fitting the quadratic to a subset of the neighboring scatter points in a weighted least squares fashion. In order for the matrix equation used to be solve for the coefficients to be stable, there should be at least ten non-coplanar scatter points in the set.

Subset Definition

In the IDW Interpolation Options dialog, an option is available for using a subset of the scatter points (as opposed to all of the available scatter points) in the computation of the nodal function coefficients and in the computation of the interpolation weights. Using a subset of the scatter points drops distant points from consideration since they are unlikely to have a large influence on the nodal function or on the interpolation weights. In addition, using a subset can speed up the computations since less points are involved.

If the Use subset of points option is chosen, the Subsets button can be used to bring up the Subset Definition dialog. Two options are available for defining which points are included in the subset. In one case, only the nearest N points are used. In the other case, only the nearest N points in each quadrant are used as shown below. This approach may give better results if the scatter points tend to be clustered.



The Four Quadrants Surrounding an Interpolation Point.

If a subset of the scatter point set is being used for interpolation, a scheme must be used to find the nearest N points. Two methods for finding a subset are provided in the Subset Definition dialog: the global method and the local method.

Global Method

With the global method, each of the scatter points in the set are searched for each interpolation point to determine which N points are nearest the interpolation point. This technique is fast for small scatter point sets but may be slow for large sets.

Local Method

With the local methods, the scatter points are triangulated to form a temporary TIN before the interpolation process begins. To compute the nearest N points, the triangle containing the interpolation point is found and the triangle topology is then used to sweep out from the interpolation point in a systematic fashion until the N nearest points are found. The local scheme is typically much faster than the global scheme for large scatter point sets.

Computation of Interpolation Weights

When computing the interpolation weights, three options are available for determining which points are included in the subset of points used to compute the weights and perform the interpolation: subset, all points, and enclosing triangle.

Subset of Points

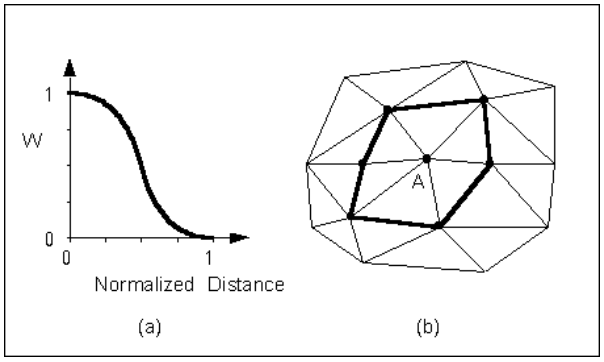
If the **Use subset of points** option is chosen, the Subset Definition dialog can be used to define a local subset of points.

All Points

If the **Use all points** option is chosen, a weight is computed for each point and all points are used in the interpolation.

Enclosing Triangle

The **Use vertices of enclosing triangle** method makes the interpolation process a local scheme by taking advantage of TIN topology (Franke & Nielson, 1980). With this technique, the subset of points used for interpolation consists of the three vertices of the triangle containing the interpolation point. The weight function or blending function assigned to each scatter point is a cubic S-shaped function as shown in part a of the figure below. The fact that the slope of the weight function tends to unity at its limits ensures that the slope of the interpolating surface is continuous across triangle boundaries.



(a) S-Shaped Weight Function and (b) Delauney Point Group for Point A.

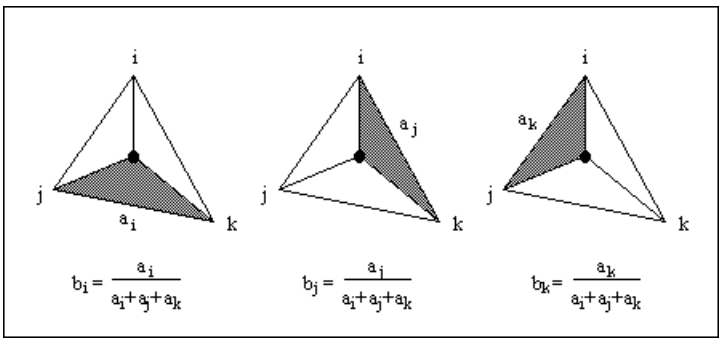
The influence of the weight function extends over the limits of the Delauney point group of the scatter point. The Delauney point group is the "natural neighbors" of the scatter point, and the perimeter of the group is made up of the outer edges of the triangles that are connected to the scatter point as shown in part b. The weight function varies from a weight of unity at the scatter point to zero at the perimeter of the group. For every interpolation point in the interior of a triangle there are three nonzero weight functions (the weight functions of the three vertices of the triangle). For a triangle T with vertices i, j, & k, the weights for each vertex are determined as follows:

$$w_i(x,y) = b_i^2\,(3-2b_i) + 3\frac{b_i^2 b_j b_k}{b_i b_j + b_i b_k + b_j b_k}\left\{ b_j\left[\frac{\|e_i\|^2 + \|e_k\|^2 - \|e_j\|^2}{\|e_k\|^2}\right] + b_k\left[\frac{\|e_i\|^2 + \|e_j\|^2 - \|e_k\|^2}{\|e_j\|^2}\right]\right\}$$

Where $\|e_i\|$ is the length of the edge opposite vertex i, and b_i, b_j, b_k are the area coordinates of the point (x,y) with respect to triangle T.

Area coordinates are coordinates that describe the position of a point within the interior of a triangle relative to the vertices of the triangle. The coordinates are based solely on the geometry of the triangle. Area coordinates are

sometimes called "barycentric coordinates." The relative magnitude of the coordinates corresponds to area ratios as shown below:



Barycentric Coordinates for a Point in a Triangle.

The XY coordinates of the interior point can be written in terms of the XY coordinates of the vertices using the area coordinates as follows:

$$x = b_i x_i + b_j x_j + b_k x_k$$
$$y = b_i y_i + b_j y_j + b_k y_k$$
$$1.0 = b_i + b_j + b_k$$

Solving the above equations for b_i, b_j, and b_k yields:

$$b_i = \frac{1}{2A}\Big[(x_j y_k - x_k y_j) + (y_j - y_k)x + (x_k - x_j)y\Big]$$
$$b_j = \frac{1}{2A}\Big[(x_k y_i - x_i y_k) + (y_k - y_i)x + (x_i - x_k)y\Big]$$
$$b_k = \frac{1}{2A}\Big[(x_i y_j - x_j y_i) + (y_i - y_j)x + (x_j - x_i)y\Big]$$
$$A = \frac{1}{2}\Big(x_i y_i + x_j y_k + x_k y_i - y_i x_j - y_j x_k - y_k x_i\Big)$$

Using the weight functions defined above, the interpolating surface at points inside a triangle is computed as:

$$F(x,y) = w_i(x,y)Q_i(x,y) + w_j(x,y)Q_j(x,y) + w_k(x,y)Q_k(x,y)$$

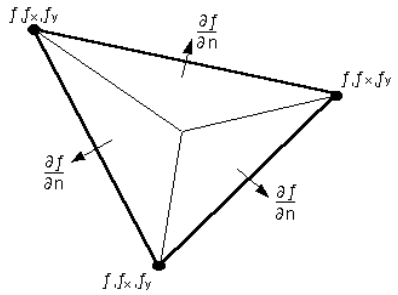
where w_i, w_j, and w_k are the weight functions and Q_i, Q_j, and Q_k are the nodal functions for the three vertices of the triangle.

4.4. Clough-Tocher

Clough-Tocher

The Clough-Tocher interpolation technique is often referred to in the literature as a finite element method because it has origins in the finite element method of numerical analysis. Before any points are interpolated, the scatter points are first triangulated to form a temporary TIN. A bivariate polynomial is defined over each triangle, creating a surface made up of a series of triangular Clough-Tocher surface patches.

The Clough-Tocher patch is a cubic polynomial defined by twelve parameters shown in the following figure: the function values, f, and the first derivatives, f_x & f_y, at each vertex, and the normal derivatives, , at the midpoint of the three edges in the triangle (Clough & Tocher, 1965; Lancaster & Salkauskas, 1986). The first derivatives at the vertices are estimated using the average slopes of the surrounding triangles. The element is partitioned into three subelements along seams defined by the centroid and the vertices of the triangle.



The Twelve Parameters Used to Define the Clough-Tocher Triangle

A complete cubic polynomial of the form:

$$F(x,y) = \sum_{j=0}^{3-i} c_{ij} x^i y^i$$

is created over each sub-triangle with slope continuity across the seams and across the boundaries of the triangle. Second derivative continuity is not maintained across the seams of the triangle.

Since the Clough-Tocher scheme is a local scheme, it has the advantage of speed. Even very large scatter point sets can be interpolated quickly. It also tends to give a smooth interpolating surface which brings out local trends in the data set quite accurately.

Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the Clough-Tocher interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the default extrapolation value entered at the bottom of the Interpolation Options dialog.

4.5. Natural Neighbor

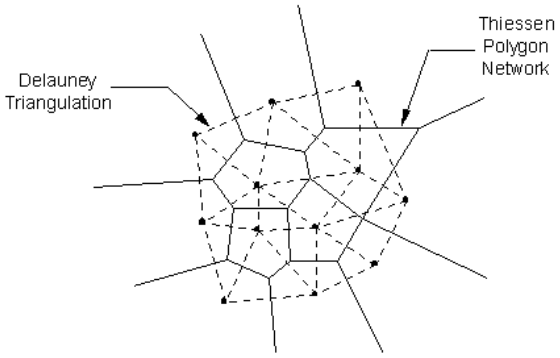
Natural Neighbor

The basic equation used in natural neighbor interpolation is identical to the one used in IDW interpolation:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i$$

As with IDW interpolation, the nodal functions can be either constants, gradient planes, or quadratics. The nodal function can be selected using the Natural Neighbor Interpolation Options dialog. The difference between IDW interpolation and natural neighbor interpolation is the method used to compute the weights and the method used to select the subset of scatter points used for interpolation.

Natural neighbor interpolation is based on the Thiessen polygon network of the scatter point set. The Thiessen polygon network can be constructed from the Delauney triangulation of a scatter point set. A Delauney triangulation is a TIN that has been constructed so that the Delauney criterion has been satisfied.



Delauney Triangulation and Corresponding Thiessen Polygon Network for a Set of Scatter Points.

There is one Thiessen polygon in the network for each scatter point. The polygon encloses the area that is closer to the enclosed scatter point than any other scatter point. The polygons in the interior of the scatter point set are closed polygons and the polygons on the convex hull of the set are open polygons.

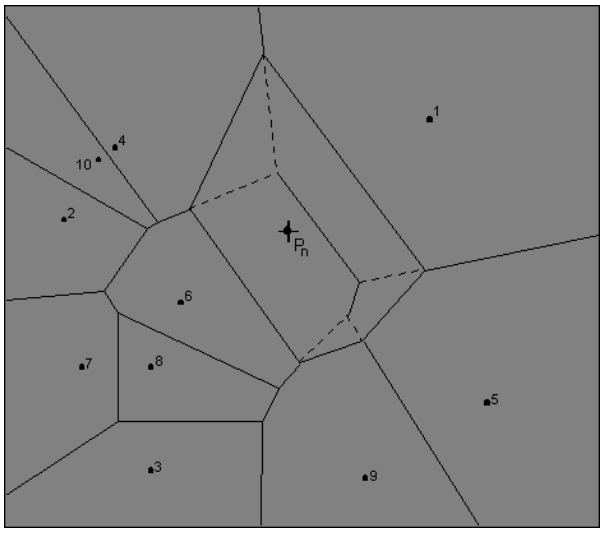
Each Thiessen polygon is constructed using the circumcircles of the triangles resulting from a Delauney triangulation of the scatter points. The vertices of the Thiessen polygons correspond to the centroids of the circumcircles of the triangles.

Local Coordinates

The weights used in natural neighbor interpolation are based on the concept of local coordinates. Local coordinates define the "neighborliness" or amount of influence any scatter point will have on the computed value at the interpolation point. This neighborliness is entirely dependent on the area of influence of the Thiessen polygons of the surrounding scatter points.

To define the local coordinates for the interpolation point, P_n, the area of all Thiessen polygons in the network must be known. Temporarily inserting P_n into the TIN causes the TIN and the corresponding Thiessen network to change, resulting in new Thiessen areas for the polygons in the neighborhood of P_n.

The concept of local coordinates is shown graphically in the following figure. Points 1-10 are scatter points and P_n is a point where some value associated with points 1-10 is to be interpolated. The dashed lines show the edges of the Thiessen network before P_n is temporarily inserted into the TIN and the solid lines show the edges of the Thiessen network after P_n is inserted.



Overlapping Thiessen Polygon Areas Used in Computation of Local Coordinates.

Only those scatter points whose Thiessen polygons have been altered by the temporary insertion of P_n are included in the subset of scatter points used to interpolate a value at P_n. In this case, only points 1, 4, 5, 6, & 9 are used. The local coordinate for each of these points with respect to P_n is defined as the area shared by the Thiessen polygon defined by point P_n and the Thiessen polygon defined by each point before point P_n is added. The greater the common area, the larger the resulting local coordinate, and the larger the influence or weight the scatter point has on the interpolated value at P_n.

If we define k(n) as the Thiessen polygon area of P_n and $k_m(n)$ as the difference in the Thiessen polygon area of a neighboring scatter point, P_m, before and after P_n is inserted, then the local coordinate $l_m(n)$ is defined as:

$$\lambda_m(n) = \frac{\kappa_m(n)}{\kappa(n)}$$

The local coordinate $l_m(n)$ varies between zero and unity and is directly used as the weight, $w_m(n)$, in the interpolation equation. If P_n is at precisely the same location as P_m, then the Thiessen polygon areas for P_n and P_m are identical and $l_m(n)$ has a value of unity. In general, the greater the relative distance P_m is from P_n, the smaller its influence on the final interpolated value.

Extrapolation

As shown in the figure above, the Thiessen polygons for scatter points on the perimeter of the TIN are open-ended polygons. Since such polygons have an infinite area, they cannot be used directly for natural neighbor interpolation. Thus, a special approach is used to facilitate extrapolation with the natural neighbor scheme. Prior to interpolation, the X and Y boundaries of the object being interpolated to (grid, mesh, etc.) are determined and a box is placed around the object whose boundaries exceed the limits of the object by approximately 10% (this value can be modified by the user). Four temporary "pseudo-scatter points" are created at the four corners of the box. The inverse distance weighted interpolation scheme with gradient plane nodal functions is then used to estimate a data value at the pseudo-points. From that point on, the pseudo-points with the extrapolated values are included with the actual scatter points in the interpolation process. Consequently, all of the points being interpolated to are guaranteed to be within the convex hull of the scatter point set. Once the interpolation is complete, the pseudo-points are discarded.

4.6. Kriging

Kriging

Kriging is a method of interpolation named after a South African mining engineer named D. G. Krige who developed the technique in an attempt to more accurately predict ore reserves. Over the past several decades kriging has become a fundamental tool in the field of geostatistics.

Kriging is based on the assumption that the parameter being interpolated can be treated as a regionalized variable. A regionalized variable is intermediate between a truly random variable and a completely deterministic variable in that it varies in a continuous manner from one location to the next and therefore points that are near each other have a certain degree of spatial correlation, but points that are widely separated are statistically independent (Davis, 1986). Kriging is a set of linear regression routines which minimize estimation variance from a predefined covariance model.

The kriging routines implemented in GMS are based on the Geostatistical Software Library (GSLIB) routines published by Deutsch and Journel (1992). Since kriging is a rather complex interpolation technique and includes numerous options, a complete description of kriging is beyond the scope of this reference manual. The user is strongly encouraged to refer the GSLIB textbook for more information:

Deutsch, C.V., & A.G. Journel. *GSLIB: Geostatistical Software Library and User's Guide*. Oxford University Press, New York, 1992.

Other good references on kriging include:

Royle, A.G., F.L. Clausen, & P. Frederiksen. Practical Universal Kriging and Automatice Contouring. *Geo-Processing*, Vol. 1, No. 4, 1981.

Davis, J.C. *Statistics and Data Analysis in Geology*. John Wiley & Sons, New York, 1986.

Lam, N.S. Spatial Interpolation Methods: A Review. *The American Cartographer*. Vol. 10, No. 2, 1983.

Heine, G.W., A Controlled Study of Some Two-Dimensional Interpolation Methods, *COGS Computer Contributions*, Vol. 2, No. 2.

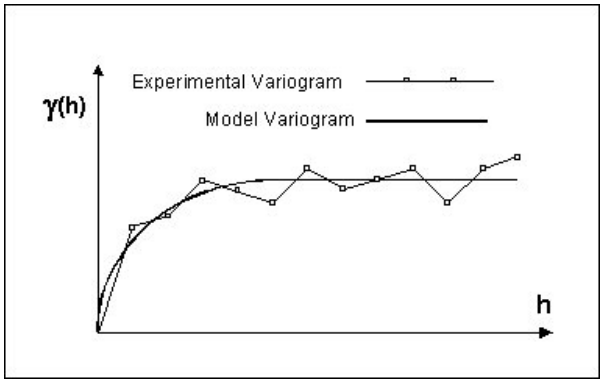
Olea, T.A., Optimal Contour Mapping using Universal Kriging. *J. Geophys. Research*, Vol. 79, No. 5, 1974.

Journel, A.G., & Huijbregts, C.J. *Mining Geostatistics*. Academic Press, New York, NY, 1978.

A powerful set of kriging techniques with varying degrees of sophistication have been implemented in GMS. The selection of the Kriging method and the definition of the variograms are accomplished using the Kriging Options dialog. There are several differences between 2D and 3D Kriging. The supported techniques include:

Ordinary Kriging

The first step in ordinary kriging is to construct a variogram from the scatter point set to be interpolated. A variogram consists of two parts: an experimental variogram and a model variogram. Suppose that the value to be interpolated is referred to as f. The experimental variogram is found by calculating the variance (g) of each point in the set with respect to each of the other points and plotting the variances versus distance (h) between the points. Several formulas can be used to compute the variance, but it is typically computed as one half the difference in f squared.



Experimental and Model Variogram Used in Kriging

Once the experimental variogram is computed, the next step is to define a model variogram. A model variogram is a simple mathematical function that models the trend in the experimental variogram.

As can be seen in the above figure, the shape of the variogram indicates that at small separation distances, the variance in f is small. In other words, points that are close together have similar f values. After a certain level of separation, the variance in the f values becomes somewhat random and the model variogram flattens out to a value corresponding to the average variance.

Once the model variogram is constructed, it is used to compute the weights used in kriging. The basic equation used in ordinary kriging is as follows:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i$$

Where n is the number of scatter points in the set, f_i are the values of the scatter points, and w_i are weights assigned to each scatter point.

This equation is essentially the same as the equation used for inverse distance weighted interpolation (equation 9.8) except that rather than using weights based on an arbitrary function of distance, the weights used in kriging are based on the model variogram. For example, to interpolate at a point P based on the surrounding points P_1, P_2, and P_3, the weights w_1, w_2, and w_3 must be found. The weights are found through the solution of the simultaneous equations:

$$w_1 S(d_{11}) + w_2 S(d_{12}) + w_3 S(d_{13}) = S(d_{1p})$$
$$w_1 S(d_{21}) + w_2 S(d_{22}) + w_3 S(d_{23}) = S(d_{2p})$$
$$w_1 S(d_{31}) + w_2 S(d_{32}) + w_3 S(d_{33}) = S(d_{3p})$$

where $S(d_{ij})$ is the model variogram evaluated at a distance equal to the distance between points i and j. For example, $S(d_{1p})$ is the model variogram evaluated at a distance equal to the separation of points P_1 and P. Since it is necessary that the weights sum to unity, a fourth equation is added:

$$w_1 + w_2 + w_3 = 1.0$$

Since there are now four equations and three unknowns, a slack variable, l, is added to the equation set. The final set of equations is as follows:

$$w_1 S(d_{11}) + w_2 S(d_{12}) + w_3 S(d_{13}) + \lambda = S(d_{1p})$$
$$w_1 S(d_{21}) + w_2 S(d_{22}) + w_3 S(d_{23}) + \lambda = S(d_{2p})$$
$$w_1 S(d_{31}) + w_2 S(d_{32}) + w_3 S(d_{33}) + \lambda = S(d_{3p})$$
$$w_1 + w_2 + w_3 + 0 = 1.0$$

The equations are then solved for the weights w_1, w_2, and w_3. The f value of the interpolation point is then calculated as:

$$f_p = w_1 f_1 + w_2 f_2 + w_3 f_3$$

By using the variogram in this fashion to compute the weights, the expected estimation error is minimized in a least squares sense. For this reason, kriging is sometimes said to produce the best linear unbiased estimate. However, minimizing the expected error in a least squared sense is not always the most important criteria and in some cases, other interpolation schemes give more appropriate results (Philip & Watson, 1986).

An important feature of kriging is that the variogram can be used to calculate the expected error of estimation at each interpolation point since the estimation error is a function of the distance to surrounding scatter points. The estimation variance can be calculated as:

$$s_z^2 = w_1 S(d_{1p}) + w_2 S(d_{2p}) + w_3 S(d_{3p}) + \lambda$$

When interpolating to an object using the kriging method, an estimation variance data set is always produced along with the interpolated data set. As a result, a contour or iso-surface plot of estimation variance can be generated on the target mesh or grid.

Simple Kriging

Simple kriging is similar to Ordinary Kriging except that the following equation is not added to the set of equations:

$$w_1 + w_2 + w_3 = 1.0$$

and the weights do not sum to unity. Simple kriging uses the average of the entire data set while ordinary kriging uses a local average (the average of the scatter points in the kriging subset for a particular interpolation point). As a result, simple kriging can be less accurate than ordinary kriging, but it generally produces a result that is "smoother" and more aesthetically pleasing.

Universal Kriging

One of the assumptions made in kriging is that the data being estimated are stationary. That is, as you move from one region to the next in the scatter point set, the average value of the scatter points is relatively constant. Whenever there is a significant spatial trend in the data values such as a sloping surface or a localized flat region, this assumption is violated. In such cases, the stationary condition can be temporarily imposed on the data by use of a drift term. The drift is a simple polynomial function that models the average value of the scatter points. The residual is the difference between the drift and the actual values of the scatter points. Since the residuals should be stationary, kriging is performed on the residuals and the interpolated residuals are added to the drift to compute the estimated values. Using a drift in this fashion is often called "universal kriging."

Kriging Options

The kriging options can be edited with the Kriging Options dialog. The options are as follows:

Kriging Method

The pull-down list in the kriging method section is used to select which kriging technique is used. The options are simple kriging or ordinary kriging. (See Kriging)

Drift

When performing Universal Kriging, a drift function should be defined. The Drift button brings up the Drift dialog. Each of the toggles in the dialog represents a single component of the polynomial equation defining the drift. Initially, all of the toggles off by default. Turning on coefficients enables universal kriging and defines the drift polynomial. For example, to use a planar drift function, only the linear terms should be used.

Search Options

The Search Options button brings up the Search Options dialog. The Minimum and Maximum values in the Number of points to use for kriging controls how many of the points found in the search radius are actually used in the kriging calculations. If fewer than the minimum value are found, a default value (-999) is assigned to the interpolation point. If greater than the maximum value is found, the closest points are used.

The input data cutoff values are used to screen out data values outside the specified range. Points with values outside this range are ignored.

If the Octant option is selected in the search type section, a maximum of N points in each of the eight octants (for 2D a quadrant is used) surrounding the interpolation point are used in the calculations. This method results in better performance with clustered data. If the Normal method is selected, the octant approach is not used.

Search Ellipsoid

The Search Ellipsoid button brings up the Search Ellipsoid dialog. When a value is interpolated to an interpolation point, only a subset of the scatter points in the vicinity of the interpolation point are used in the calculations. The items in the Search Ellipsoid dialog control the shape of a "search space" surrounding the interpolation point. Only points in this search space are considered candidates for use in the kriging calculations.

By default, the search space is a circle (sphere in 3D) centered at the point with a radius defined by the Maximum search radius item. For problems exhibiting anisotropy, the search space can be transformed to an ellipse (ellipsoid in 3D). The anis1 factor and the azimuth angle control the shape and orientation of the ellipse. The azimuth represents the rotation of the major principal axis clockwise from the +y axis. The anis1 factor represents the ratio of the search radius along the minor principal axis relative to the search radius (the maximum radius) in the major principal direction. In most cases, the anis1 factor and the azimuth angle should match the anis factor and azimuth angle defined in the Variogram Editor.

Editing Variograms

Regardless of which kriging method is selected, a model variogram must be constructed prior to interpolating the values from the scatter points to the target object. In some cases, multiple variograms must be defined. The basic steps involved in constructing a model variogram are to first build an experimental variogram and then construct a model variogram that matches the experimental variogram. Variograms are constructed using the Variogram Editor. The Variogram Editor is activated by selecting the Edit Variogram button in the Kriging Options dialog.

3D Kriging

3D Kriging is almost identical to 2D Kriging. All of the basic kriging options, including simple kriging and ordinary kriging. (See Kriging)

2D vs. 3D

There are several differences in the 2D and the 3D versions of kriging. First of all, if the drift option is turned on, more drift coefficients are available. In the Search Options dialog, an octant searching scheme can be selected. A number is entered which represents the maximum number of scatter points from each of the eight octants surrounding the interpolation point to keep in the subset. Limiting the number of points in each octant can give better results when the scatter points are clustered.

Modeling Anisotropy

The main difference between the 3D and 2D versions of kriging is the way anisotropy is treated. The third dimension adds additional angles and factors that must be manipulated. As is the case with 2D kriging, the first step in modeling anisotropy is to detect anisotropy using experimental variograms. Anisotropy can be modeled in up to three orthogonal directions. A series of orthogonal variograms are generated at different orientations until the three experimental variograms corresponding to the three principal axes of anisotropy are found. The combination which gives the greatest difference in range for the three experimental variograms corresponds to the principal axes. The axis with the largest range is the major principal axis.

When computing directional experimental variograms in 3D, two angles are used to define the direction vector: azimuth and dip. To define the rotation of a vector, we assume the unrotated vector starts in the +y direction. The azimuth angle is the first angle of rotation and it represents a clockwise rotation in the horizontal plane starting from the +y axis. The dip angle is the second angle of rotation and it represents a downward rotation of the vector from the horizontal plane. The azimuth and dip angles defined in the experimental variogram dialog can be used to define a focused experimental variogram in any direction.

Once anisotropy has been detected using the experimental variograms, anisotropy can be modeled with the model variogram using either the directional variogram method or the anisotropy factor method. The simplest method is the directional variogram approach. If the directional variogram approach is used, a separate model variogram is constructed for each of the three orthogonal axes.

If the anisotropy factor method is selected, the azimuth and dip angles corresponding to the major principal axis should be entered into the angle edit fields in the lower left corner of the Variogram Editor. These fields also allow a third angle of rotation, the plunge angle, to be specified. The plunge angle represents a rotation or spinning about the direction vector (which is already rotated by the azimuth and dip). The direction of rotation is defined as clockwise looking down the direction vector toward the origin. In most cases, the plunge angle can be left at zero.

Once the angles are entered, the model variogram should then be constructed which fits the experimental variogram corresponding to the major principal direction. The anis1 and anis2 parameters in the Variogram Editor should then be changed to a value other than unity (the default value). Changing these parameters to a value less than unity causes three curves to be drawn for the model variogram. The second curve corresponds to the original curve with the range parameter multiplied by the anis1 value. The third curve corresponds to the original curve with the range parameter multiplied by the anis2 value. The anis1 parameter should be altered until the second curve fits the experimental variogram corresponding to the second principal axis of anisotropy. If the principal axis is assumed to be the y axis in the unrotated state, this axis is the x axis in the rotated state. The anis2 parameter should then be altered until the third curve matches the third principal axis of anisotropy (the z axis in the unrotated state). Once the correct anisotropy factors are found, the Variogram Editor should be exited and the angles and anisotropy factors

should be entered in the Search Ellipsoid dialog to define a search ellipsoid that matches the variogram anisotropy.

For further information on modeling anisotropy in 3D, the user is referred to Deutsch and Journel (1992).

Variogram Editor

Before interpolating a scatter point set using the Kriging option, a model variogram must be defined. The basic steps involved in constructing a model variogram are to first build an experimental variogram and then construct a model variogram that matches the experimental variogram. This is accomplished using the Variogram Editor. The Variogram Editor is activated by selecting the Edit Variogram button in the Kriging Options dialog.

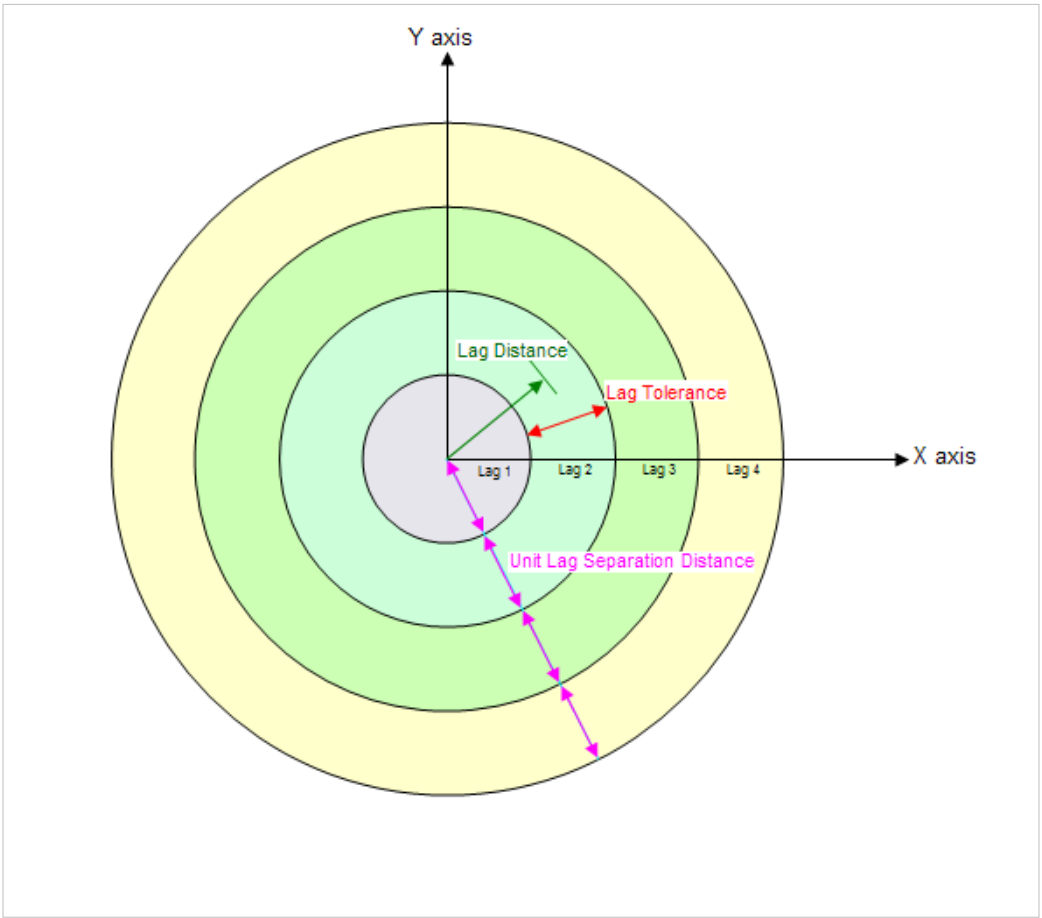
The experimental variograms and the model variogram are plotted in the upper portion of the Variogram Editor. The items in the upper right portion of the Editor are used to create experimental variograms. The items in the lower half of the Editor are used to define the model variogram. In a typical study, several experimental variograms may be constructed and plotted before one is chosen. A model variogram is then designed to fit the chosen experimental variogram.

Creating Experimental Variograms

A new experimental variogram is computed by selecting the New button under the list of experimental variograms. This button brings up the Experimental Variogram dialog.

Lags

When computing an experimental variogram, it is impractical to plot a variance for each scatter point with respect to each of the other scatter points. Therefore, distances are subdivided into a number of intervals called lags as illustrated in the following figure. The distance between each pair of scatter points is checked to see which lag interval it lies within. The variances for all pairs of points whose separation distance falls within the same lag interval are averaged. The resulting average is plotted in the experimental variogram vs. the distance corresponding to the lag interval. Therefore, there is one point in the experimental variogram plot for each lag. The lag intervals are defined in the Experimental Variogram dialog by entering a total number of lags, a unit lag separation distance, and a lag tolerance. In most cases, the lag tolerance should be one half of the unit lag separation distance; the tolerance can be smaller to allow for data on a pseudo-regular grid. A pair will be included in multiple lags if the tolerance is greater than one-half of the unit lag separation distance.

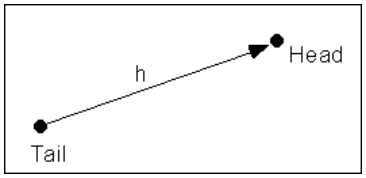


Semivariogram

The semivariogram is the most common type of variogram. The semivariogram value for a lag interval is computed as:

$$\gamma(h) = \frac{1}{2N}\sum_{i=1}^{N}\left(f_{1i} - f_{2i}\right)^2$$

where N is the number of pairs of points whose separation distance falls within the lag interval and f_{1i} and f_{2i} are the values at the head and tail of each pair of points. The head and tail are defined as follows:



Naming Convention for Pairs of Scatter Points.

Covariance

The covariance is the traditional covariance used in statistics. The covariance value for a lag interval is computed as:

$$C(h) = \frac{1}{N}\sum_{i=1}^{N}\left(f_{2i}f_{1i} - m_{-h}m_{+h}\right)$$

where m_{-h} and m_{+h} are the mean of the head and tail values respectively.

Correlogram

The correlogram is computed by standardizing the covariance by the standard deviation of the head and tail values.

$$\rho(h) = \frac{C(h)}{\sigma_{-h}\sigma_{+h}}$$

where s_{-h} and s_{+h} are the standard deviation of the head and tail values respectively.

General Relative Semivariogram

This variogram is computed by standardizing the semivariogram computed using equation 9.38 by the squared mean of the data values in each lag:

$$\gamma_{GR}(h) = \frac{\gamma(h)}{\left(\frac{m_{-h} + m_{+h}}{2}\right)^2}$$

Pairwise Relative Semivariogram

With this variogram, each pair is normalized by the squared average of the tail and head values.

$$\gamma_{PR}(h) = \frac{1}{2N}\frac{(f_{1i} - f_{2i})^2}{\left[\frac{(f_{1i} - f_{2i})^2}{2}\right]^2}$$

Experience has shown that the general relative and pairwise relative semivariograms are effective in revealing spatial structure and anisotropy when the scatter points are sparse (Deutsch & Journel, 1992). Because of the divisors in equations 9.41 and 9.42, these semivariograms should only be used on positively skewed datasets.

Semivariogram of Logarithms

This variogram is computed by applying equation 9.38 to the natural logarithms of the data values:

$$\gamma_L(h) = \frac{1}{2N}\sum_{i=1}^{N}\left(ln(f_{1i}) - ln(f_{2i})\right)^2$$

Semirodogram

The semirodogram is similar to the traditional semivariogram except that the square root of the absolute difference is used rather than the squared difference:

$$\gamma_R(h) = \frac{1}{2N}\sum_{i=1}^{N}\sqrt{|f_{1i} - f_{2i}|}$$

Semimadogram

The semimadogram is similar to the traditional semivariogram, except that the absolute difference is used rather than the squared difference:

$$\gamma_M(h) = \frac{1}{2N}\sum_{i=1}^{N}|f_{1i} - f_{2i}|$$

The semirodogram and the semimadogram are particularly effective for establishing range and anisotropy. They should not be used for modeling the nugget of semivariograms (Deutsch & Journel, 1992).

Viewing the Experimental Variograms

After setting up the lag interval and choosing a variogram type, the OK button is selected in the Experimental Variogram dialog. At this point, the experimental variogram is computed. For large scatter point sets, this may take a significant amount of time.

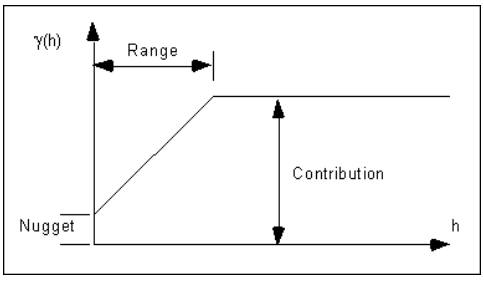
Once the experimental variogram is computed, it is added to the list of experimental variograms in the upper right corner of the Variogram Editor and it is displayed in the variogram plotting window. One of the variograms in the list is always highlighted. The name, color, and symbols (used to plot the variogram) of the highlighted variogram can be edited. In addition, the display of each variogram can be turned on and off so any combination of experimental variograms can be plotted. Selecting the Delete button deletes the highlighted variogram. Selecting the Edit button causes the Experimental Variogram dialog to come up initialized with the values used in the computation of the highlighted variogram. When the OK button is selected, the values of the variogram are recomputed.

Creating Model Variograms

Once a set of experimental variograms are computed, one is chosen and a model variogram is constructed to fit the experimental variogram. The model variogram is constructed using the items in the lower half of the Variogram Editor.

Model Functions

Four types of model functions are supported for building model variograms. Each of the functions are characterized by a nugget, contribution, and range.



The Parameters Used to Define a Model Variogram.

The nugget represents a minimum variance. The contribution is sometimes called the "sill" and represents the average variance of points at such a distance away from the point in question that there is no correlation between the points. The range represents the distance at which there is no longer a correlation between the points.

The four model functions supported are:

Spherical Model

The Spherical Model is defined by a range -a- and a contribution -c- as:

$$\gamma(h) = \begin{cases} c\left[1.5\frac{h}{a} - 0.5\left(\frac{h}{a}\right)^3\right], & \text{if } \frac{h}{a} \leq a \\ c & \text{if } \frac{h}{a} > a \end{cases}$$

Exponential Model

The Exponential Model is defined by a parameter -a- and a contribution -c- as:

$$\gamma(h) = c\left[1 - \exp\left(-\frac{3h}{a}\right)\right]$$

Gaussian Model

The Gaussian Model defined by a parameter -a- and a contribution -c- as:

$$\gamma(h) = c\left[1 - \exp\left(-\frac{3h^2}{a^2}\right)\right]$$

Power Model

The Power Model is defined by a power 0 < a < 2 and a slope c as:

$$\gamma(h) = ch^a$$

Nested Structures

A model variogram is constructed using a combination of one or more model functions. Each instance of a model function is called a "nested structure". A nested structure is created by selecting the New button in the Nested Structure section of the dialog. A new structure is created and added to the list of nested structures. The model variogram plotted in the variogram plot window represents the combination of all of the nested structures in the list. One of the nested structures in the list is highlighted at all times. The selected structure can be deleted by selecting the *Delete* button under the list. The name, model function type, contribution, and range of the selected structure can be edited (the nugget is the same for all nested structures, i.e., only the contribution and range of each structure are summed). As the parameters defining the structure are altered by the user, the plot of the model variogram is updated dynamically in the variogram plot window. This type of instantaneous feedback provides a powerful tool for "sculpting" a model variogram in an intuitive manner until it fits the selected experimental variogram.

In most cases, a single nested structure is adequate. For cases with complex experimental variograms, using multiple nested structures to define the model variogram can prove useful.

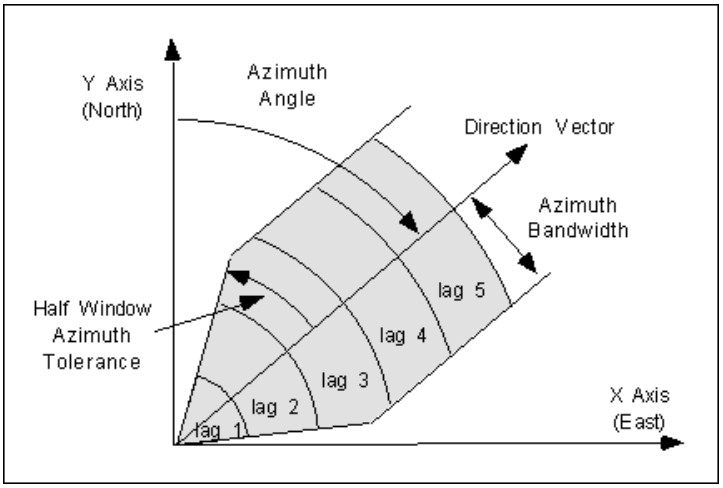
Modeling Anisotropy

Some datasets exhibit anisotropy, i.e., the correlation between scatter points changes with direction. For example, due to the depositional history of an alluvial soil deposit, parameters such as porosity and hydraulic conductivity may be most strongly correlated in one direction. This means the differences in the data values change relatively little in one direction compared to how much they change with distance in the orthogonal direction. The direction corresponding to the highest correlation (smallest change) is called the major principal direction and the orthogonal direction is the minor principal direction.

One of the more powerful features of the kriging method is that anisotropy can be detected by generating experimental variograms in orthogonal directions and looking for differences. When anisotropy exists, the model variogram can be constructed to match the anisotropy and ensure that the differences in the continuity of the data each of the orthogonal directions is accurately modeled in the interpolated dataset.

Detecting Anisotropy

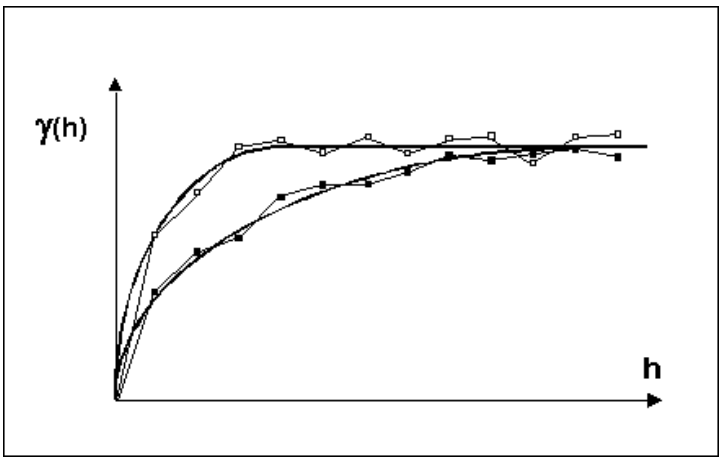
Anisotropy can be detected by generating a focused experimental variogram in each orthogonal direction and observing whether or not there are significant differences in the resulting variograms. When constructing an experimental variogram with the Experimental Variogram dialog, directional data corresponding to an axis of anisotropy can be entered. The meaning of the directional data is illustrated in the following figure:



The Directional Data Used to Detect Anisotropy.

When a scatter point is compared with each of the other scatter points to compute the experimental variogram, only those points falling within the shaded area shown in the figure above are considered. The shaded area is defined by the azimuth angle, the azimuth bandwidth, the half window azimuth tolerance, and the lag intervals. For isotropic conditions, the half window azimuth tolerance should be set to 90^o (the default value). This forces all points to be included in the calculation of the experimental variogram.

Anisotropy is typically detected using a trial and error process. Pairs of experimental variograms are generated, the pairs being offset from each other by an azimuth angle of 90^o. If anisotropy exists, the ranges of the two variograms will differ as shown below. If the data are isotropic, the azimuth angle will have little effect on the resulting experimental variograms. The angles which produce the pair of experimental variograms with the largest difference in ranges represent the principal axes of anisotropy. The variogram with the larger range represents the major principal axis and the variogram with the shorter range represents the minor principal axis.



Experimental and Model Variograms for Anisotropic Conditions.

Anisotropy Method

Once anisotropy has been detected, the next step is to model the anisotropy using the model variogram.

The azimuth angle corresponding to that major principal axis (the one with the longer range) should be entered in the azimuth angle field in the lower left corner of the Variogram Editor (the dip and plunge fields are for 3D kriging and are dimmed for 2D interpolation). A model variogram should then be constructed which fits the experimental variogram corresponding to the major principal direction. The anis1 parameter in the Variogram Editor should then be changed to a value other than unity (the default value). Changing the anis1 parameter to a value less than unity causes two curves to be drawn for the model variogram as shown in the above figure. The second curve corresponds to the original curve with the range parameter multiplied by the anis1 value. In other words, the anis1 parameter represents the range in the minor direction divided by the range in the major direction. The anis1 parameter should be altered until the second curve fits the experimental variogram corresponding to the minor principal axis of anisotropy. Each of the nested structures has an anis1 parameter that can be edited. Once again, as the anis1 parameter is altered, the variogram plot is updated dynamically, allowing a fit to be made in a simple intuitive fashion. Once the correct anis1 factor is found, the Variogram Editor should be exited and the azimuth and anis1 factors should be entered in the Search Ellipsoid dialog to define a search ellipse that matches the variogram anisotropy.

Saving Variograms

Once a variogram or set of variograms is defined, the variograms are saved with the dataset files when the project is saved to disk. Thus, when the project is read back in to GMS, the variograms are ready to be used for interpolation and do not need to be redefined.

4.7. Jackknifing

Jackknifing

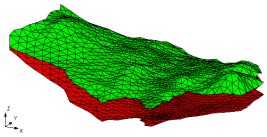
Jackknifing is a special type of interpolation which can be useful in analyzing a scatter point set or an interpolation scheme. When the ***Jackknifing*** command is selected, the active scatter point set is interpolated "to itself" using the currently selected interpolation scheme. Each point in the set is processed one at a time. The point is temporarily removed and the selected interpolation scheme is used to interpolate to the location of the missing point using the remaining points. Ideally, the interpolated value should correspond closely to the original measured value at the point. By interpolating to each point, a new data set is generated for the scatter point set. This new dataset can be compared with the original dataset using the ***Summary*** command in the *Interpolation* menu.

You can select your original data set and then you can select the dataset created from jackknifing. The mean error, mean absolute error, and the root mean squared error are automatically calculated.

5. Modules

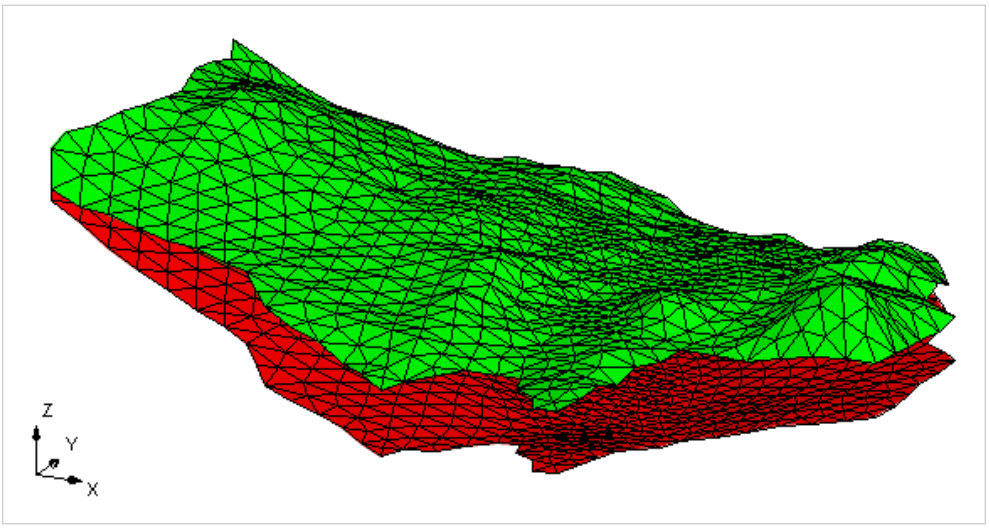
5.1. TIN Module

TIN Module

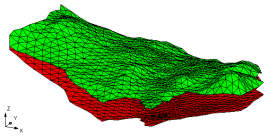
TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

TIN stands for Triangulated Irregular Network. TINs are used for surface modeling. TINs are formed by connecting a set of XYZ points with edges to form a network of triangles. TINs can be used to represent the surface of a geologic unit or the surface defined by a mathematical function.

Several TINs can be modeled at once in GMS. One of the TINs is designated as the "active" TIN. The selection and editing tools apply to the active TIN only.



Creating a TIN

TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

In order to create a TIN in GMS you must have a set of TIN vertices. Then the TIN is created by triangulating the vertices (connecting the vertices with lines to form triangles). The triangulation algorithm assumes that each of the vertices being triangulated is unique in the xy plane, i.e., no two points have the same xy location. Duplicate points can be removed by selecting Find Duplicates from the TINs menu.

TINs can be created 3 different ways in GMS: manually entering the vertex locations and triangulating, converting a different GMS data type to a TIN, and copying a currently existing TIN.

Manually Creating a TIN

A TIN can be created manually from the following steps:

1. Right-click in the empty space of the Project Explorer and select the ***New → TIN*** command.
2. Select the ***Create Vertices*** tool from the TIN Tool Palette.
3. Create the vertices by clicking inside the Graphics Window at the xy coordinates where you want the vertex located. (To change the vertex location see: Editing a TIN)
4. Select the ***Triangulate*** command from the *TINs* menu.

Creating a TIN from GMS Data

2D meshes, 2D grids, and 2D scatter points can all be converted to a TIN. This is accomplished by using the following commands:

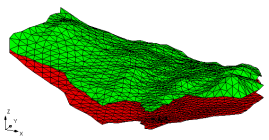
- Mesh to TIN
- Grid to TIN
- Scatter Points to TIN
- Contacts to TIN
- Watertable to TIN
- ***Add Contacts to TIN*** - This command is used to enter a point from a contact into the active TIN . The contact(s) are first selected and the command is then chosen from the *Borehole* menu. Typically all contacts which should be part of a TIN are selected before generating the TIN, but sometimes one is inadvertently left out, or more boreholes are added later.

Copying a Current TIN

To make a copy of a TIN that currently exists in GMS follow these steps:

1. Select the TIN you wish to copy using the ***Select TINs*** tool.
2. Select the ***Duplicate TIN*** command from the *TINs* menu. A dialog appears prompting for the Z offset of the new TIN. The Z offset is used to displace the TIN above or below the TIN being duplicated.

Editing a TIN

TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

TINs can be edited several ways. The selection and editing tools apply only to the active TIN. If you are going to edit vertices, you must first turn off the ***TINs | Lock All Vertices*** menu command.

Editing TIN Vertices

Creating New TIN Vertices

New vertices can be created using the ***Create Vertices*** tool from the TIN Tool Palette. Clicking in the Graphics Window creates a new vertex at the point clicked (vertices can only be created when in ***Plan View***). The default z value and other parameters governing the creation of new vertices can be set by selecting the ***TIN Settings*** command from the *TINs* menu.

Deleting TIN Vertices

Selected TIN vertices can be deleted by hitting the **Delete key** or by selecting the ***Delete*** command from the *Edit* menu. If the ***Confirm Deletions*** option in the Preferences dialog is on, the user is prompted to confirm each deletion.

Editing TIN Vertex Coordinates

Two methods of editing TIN vertex coordinates are available. To manipulate vertex coordinates, the ***Select Vertex*** tool must be selected from the TIN Tool Palette.

- A vertex can be moved to a new position by clicking on the vertex and holding down the mouse button while dragging the vertex to the desired position. If the current view is plan view, dragging the vertex causes it to move in the xy plane. GMS does not allow the vertex to be dragged to a position where one of the surrounding triangles becomes inverted. If the current view is not the plan view, the vertex moves along the z-axis.

- The vertex position and z value can also be manipulated by selecting the vertex and changing the XYZ values that will appear in the x, y, and z edit boxes in the Edit Window.

Snap Vertices to TIN

It is sometimes useful to snap the vertices of one TIN to another TIN. This is useful when modeling pinch out zones and truncations. The TIN containing the vertices to be moved should be the active TIN, since vertex selection can only be done for the active TIN. After the desired vertices have been selected, the ***Snap Vertices to TIN*** command of the *TINs* menu should be selected. GMS then prompts the user to select the TIN to which the vertices are snapped. The selected vertices' z coordinate values are then modified such that they lie on the selected TIN.

Editing Triangles of a TIN

Create Triangles

The ***Create Triangles*** tool is used to manually create new triangles. Triangles are normally created by triangulating a set of points automatically. However, this tool is useful for manually editing and refining a TIN. To use the Create Triangles tool:

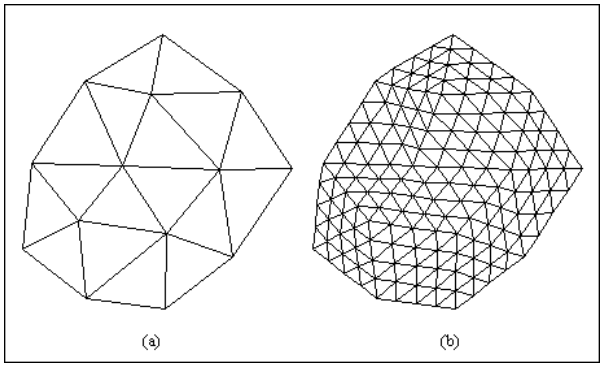
- Select the three vertices of the triangle. The vertices can be selected in either clockwise or counter-clockwise order.
- Drag a box around three vertices of the triangle.

Deleting Triangles

- Using the ***Select Triangles*** tool, the triangles may be selected and deleted.
- **Boundary Triangles –** The perimeter of the TIN resulting from the triangulation process corresponds to or approximates the convex hull of the TIN vertices. This may result in some long thin triangles or "slivers" on the perimeter of the triangulated region. There are several ways to deal with the long thin triangles. Thin triangles can be selected and deleted using the normal selection procedures. There is also an option for selecting thin triangles when the Select Triangles tool is selected. If the **Control key** is held down, it is possible to drag a line with the mouse. All triangles intersecting the line are selected. Long thin triangles on the perimeter of the TIN can also be selected by selecting the ***Select Boundary Triangles*** command from the *TINs* menu. The ***Select Boundary Triangles*** command checks triangles on the outer boundary first. If the length ratio of the triangle is less than the critical length ratio, the triangle is selected and the triangles adjacent to the triangle are then checked. The process continues inward until none of the adjacent triangles violate the minimum length ratio. The critical length ratio for selecting thin triangles can be set by selecting the ***TINs | TIN Settings*** menu command. The length ratio is defined as the longest side of the triangle divided by the sum of the two shorter sides.

Changing Triangle Density

The density of a TIN can be quickly increased using the ***Uniformly Subdivide TIN*** command in the *TINs* menu. The user is prompted for a subdivision factor and the factor is used to uniformly subdivide the TIN into sub-triangles as shown below:



TIN (a) Before and (b) After Uniform Subdivision.

Subdivide TIN

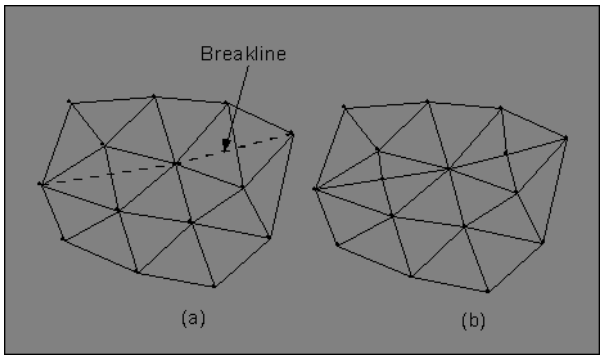
This command can be used to "smooth" a TIN. When using a TIN for contouring, the contours are computed using a linear interpolation of the triangles. If the vertices are sparse, the contours may not appear to be smooth. The contours can be smoothed by copying the vertices to a scatter point set, subdividing the TIN into sub-triangles, and interpolating the z values (or other data sets) from the scatter point set to the new vertices defining the sub-triangles.

Subdivision and smoothing can be accomplished using the following steps:

- If multiple TINs exist, make sure the TIN is the active TIN.
- Convert the TIN to a scatter point set using the ***TIN -> Scatter Points*** command in the *TINs* menu.
- Subdivide the TIN by selecting the ***Subdivide TIN*** command from the *TINs* menu.
- Switch to the 2D Scatter Point module and select an interpolation method using the ***Interp. Options*** command in the *Interpolation* menu.
- Select the to ***Active TIN*** command from the *Interpolation* menu. This creates a new data set for the selected TIN.

Adding Breaklines

A breakline is a feature line or polyline representing a ridge or some other feature that the user wishes to preserve in a mesh made up of triangular elements. In other words, a breakline is a series of edges to which the triangles should conform to, i.e., not intersect.

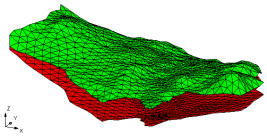


Breaklines (a) Triangulated Mesh and Breakline. (b) Triangulated Mesh After the Breakline has been Processed.

Breaklines can be processed using the ***Add Breaklines*** command from the **Mesh** menu. Before selecting the command, one or more sequences of nodes defining the breakline(s) should be selected using the ***Select Node Strings*** tool in the 2D Mesh Tool Palette.

As each breakline is processed, the triangles intersected by the breakline are modified by adding new nodes at necessary locations to ensure that the edges of the triangles will conform to the breakline. The elevations of the new nodes are based on a linear interpolation of the breakline segments. The locations of the new nodes are determined in such a way that the Delauney criterion is satisfied.

TIN Settings

TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

Retriangulate After Deleting – If this option is on, the region surrounding the vertex is retriangulated as each vertex is deleted. Otherwise, the triangles adjacent to the vertex are simply deleted.

Adjust Boundary to Include Exterior Vertices – If this option is on, the boundary of the TIN is changed so that the new vertex becomes part of the TIN if a new point is added outside the active TIN. If the new vertex is in the interior of the active TIN, the vertex is automatically incorporated into the TIN.

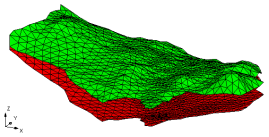
Default Z-Value – The default z value is assigned to all new vertices created with the Create Vertex tool.

Confirm Z-Values – If this option is on, GMS prompts for a z value each time a new vertex is created.

Interpolate For Default Z On Interior – If this option is on and a new vertex is created in the interior of a TIN, a default z-value is linearly interpolated from the plane equation defined by the triangle containing the point.

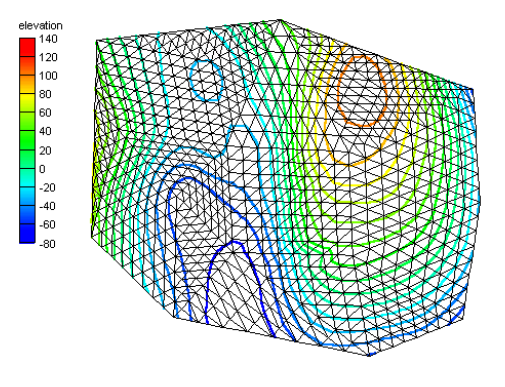
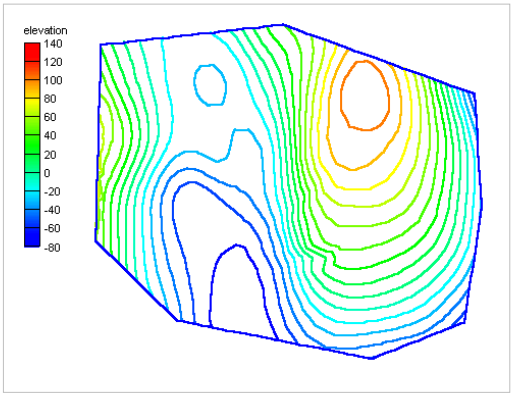
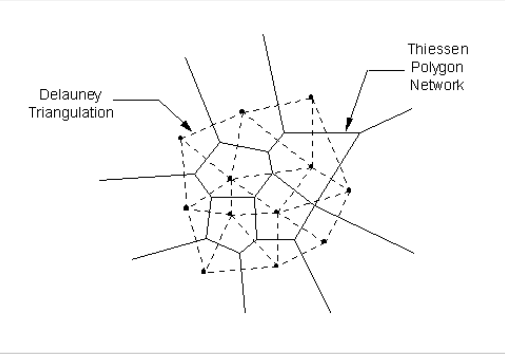
Extrapolate For Default Z On Exterior – If this option is on and a new vertex is entered outside the TIN boundary, a default z-value is extrapolated from the TIN to the new vertex.

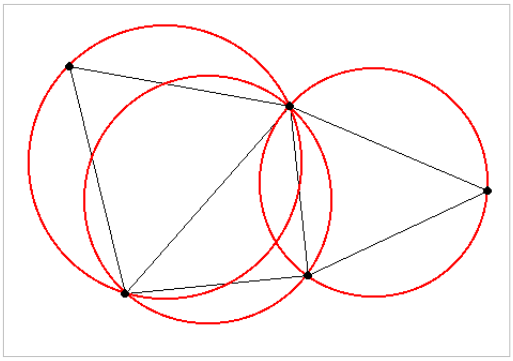
TIN Display Options

TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

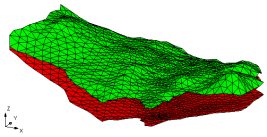
The properties of all TIN data that GMS displays on the screen can be controlled through the TIN tab of the Display Options dialog. This dialog is opened by right clicking on the TIN Data entry in the Project Explorer and selecting the Display Options command. It can also be accessed from the from the Display menu or the The following table describes the display options available for the TIN module.

Display Option	Description
Vertices	If the Vertices item in the TIN Display Options dialog is set, the TIN vertices are displayed each time the Graphics Window is refreshed. Since it is possible to accidentally drag points, vertices can be "locked" to prevent them from being dragged or edited by selecting the Lock ALL Vertices command from the TINs menu. Vertices can be unlocked by unchecking the Lock ALL Vertices command in the TINs menu. Both a "Locked" and "Unlocked" vertex color may be set so that there is a visible difference when displaying the TIN. (See Editing TINs)
Triangle edges	If this item is on the lines that make up each triangle are displayed. The color of the triangle edges can be adjusted according to the following options: 1. Auto – draws the material color if faces are not displayed. Uses black or white if the faces are displayed. 2. Specified – used the color specified next to the triangle edges 3. Material – displays the material color of the triangle
Triangle faces	The Triangle faces item causes the faces of the triangles to be drawn as filled polygons.
Texture map image to active TIN	The Texture map image item is used to "drape" an image over the surface of the TIN.

TIN boundary	The TIN boundary feature is often used in conjunction with contours in order to display the contours without cluttering the screen by displaying each triangle. The first image below shows contours displayed together with the TIN triangles. The second image shows contours displayed with the TIN boundary.  
Thiessen polygons	If the Thiessen polygons item is set, a Thiessen polygon for each TIN vertex is displayed each time the display is refreshed. The edges of the Thiessen polygons are formed by the perpendicular bisectors of the edges of the triangles in the TIN. The vertices of these polygons correspond to the centers of the circumcircles of the Delauney triangulation. Any location inside a Thiessen polygon is closer to the TIN vertex contained in that polygon than to any other TIN vertex. The red polygons in the image below correspond to the Thiessen polygons for this TIN. 

Circumcircles	If the Circumcircles item is set, the circumcircle enclosing the three vertices for each triangle are drawn when the display is refreshed. Circumcircles provide the basis of a Delauney triangulation since the Delauney criterion is satisfied by ensuring that no circumcircle encloses a vertex. Displaying circumcircles can aid in the understanding of the triangulation process. The red circles in the figure are the circumcircles for the TIN. 
Vertex numbers	If the Vertex numbers item is set, the number of each vertex is displayed adjacent to the vertex.
Scalar values	If the Scalar values item is set, the active data set value of each vertex is displayed adjacent to the vertex.
Contours	Most of the objects supported by GMS can be contoured by turning on the Contour Options in the Display Options dialog. When an object is contoured, the values associated with the active data set for the object are used to generate the contours.
Vectors	If the Vectors item in the Display Options dialog is selected for an object (TIN, Grid, or Mesh), vector plots can be generated using the active vector data set for the object. One vector is placed at each node, cell, or vertex.

TIN Tool Palette

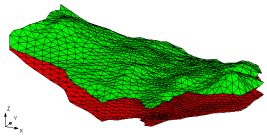
TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

The following tools appear in the dynamic portion of the Tool Palette when the TIN module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window with the cursor depends on the current tool. The tools are for selection and interactive editing of TINs. The table below describes the tools in the TIN tool palette.

Tool	Tool Name	Description
	Select Vertices	The Select Vertices tool is used to select vertices for operations such as deletion, or to drag a vertex to a new location. The coordinates of selected vertices can also be edited using the Edit Window.
	Select Triangles	The Select Triangles tool is used to select triangles for operations such as deletion.
	Select TINs	The Select TINs tool is used to select TINs for operations such as deletion. When this tool is active, a TIN icon appears at the centroid of each TIN. A small letter "A" appears in the icon of the active TIN. A TIN is selected by selecting the icon. A TIN can be designated as the active TIN by double-clicking on the TIN icon. When a different tool is selected, the icons disappear. In some cases, several TINs occupy approximately the same location and the icons for the TINs overlap. In such cases, it may be difficult to select the desired TIN. An alternate way to select TINs is to use the Select From List command in the Edit menu. This brings up a list of the currently available TINs and a TIN is selected by highlighting the name of the desired TIN and selecting the OK button.
	Select Vertex Strings	The Select Vertex Strings tool is used to select one or more strings of vertices. Vertex strings are used for operations such as adding breaklines to the TIN. The procedure for selecting vertex strings is somewhat different than the normal selection procedure. Strings are selected as follows: • Click on the starting vertex for the string. The vertex selected will be highlighted in red. • Click on any subsequent vertices you would like to be part of the string (vertices do not have to be next to each other) and double-click on the final vertex. The selected vertices are now connected by a solid red line. To remove the last vertex from a string, press the Backspace key. To abort entering a vertex string, press the ESC key. To end a vertex string, press Return or double-click on the last vertex in the string. Another vertex string can then be selected.

	Create Vertices	The Create Vertices tool is used to manually add vertices to a TIN. It can only be used in plan view. When this tool is selected, clicking on a point within the Graphics Window will place a vertex at that point. What happens to the vertex after it is added (whether and how it is triangulated into the TIN) depends on the settings in the Vertex Options dialog under the Modify TINs menu.
	Create Triangles	The Create Triangles tool is used to manually create new triangles. Triangles are normally created by triangulating a set of points automatically. However, this tool is useful for manually editing and refining a TIN. To use the Create Triangles tool: • Select the three vertices of the triangle. The vertices can be selected in either clockwise or counter-clockwise order. • Drag a box around three vertices of the triangle.
	Swap Edges	The Swap Edges tool swaps the common edge of two adjacent triangles. To use the tool, simply click on any edge in the TIN.
	Contour Lables	The Contour Label tool manually places numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contouring options are changed, until they are deleted using the Contour Label Options dialog, or until the Graphics Window is refreshed. Contour labels can be deleted with this tool by holding down the Shift key while clicking on the labels. This tool can only be used when the TIN is in plan view.

Converting TINS to Other Data Types

TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

TINs may be converted to other types of data used in GMS, such as a 2D mesh or 2D scatter points. TINs can be converted by right-clicking on the TIN in the Project Explorer, Right-clicking on the TIN in the graphics window, or using the following commands in the TIN menu:

TIN → 2D Scatter Points

The ***TIN → 2D Scatter Points*** command creates a 2D scatter point set from the active TIN . One scatter point is created for each vertex in the TIN. A copy is made of each of the datasets associated with the TIN and the duplicate data sets are stored with the new scatter point set.

TIN → 2D Mesh

The ***TIN → 2D Mesh*** command creates a 2D finite element mesh from the active TIN . One triangular element is created for each triangle in the TIN. Any datasets associated with the TIN are copied to the new mesh.

Fill Between TINs → 3D Mesh

See Creating a 3D Mesh.

TIN Boundary → Polygons

This command creates one or more polygons in the active coverage in the Map module corresponding to the outer boundary of the active TIN.

TIN Thiessen → Polygons

This command calculates the thiessen polygons from the TIN and converts them to feature polygons.

Vertex Strings → Arcs

The ***Vertex Strings → Arcs*** command creates an arc in the active coverage of the Map module for each of the selected vertex strings.

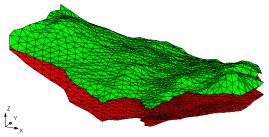
Horizons → Solids

See Horizons → Solids.

Horizons → 3D Mesh

See Horizons → 3D Mesh.

Building Solids and 3D Meshes with TINs

TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

TINs can be used to build 3D solid models as well as 3D meshes. This can be done by selecting the following commands in the Build TIN menu:

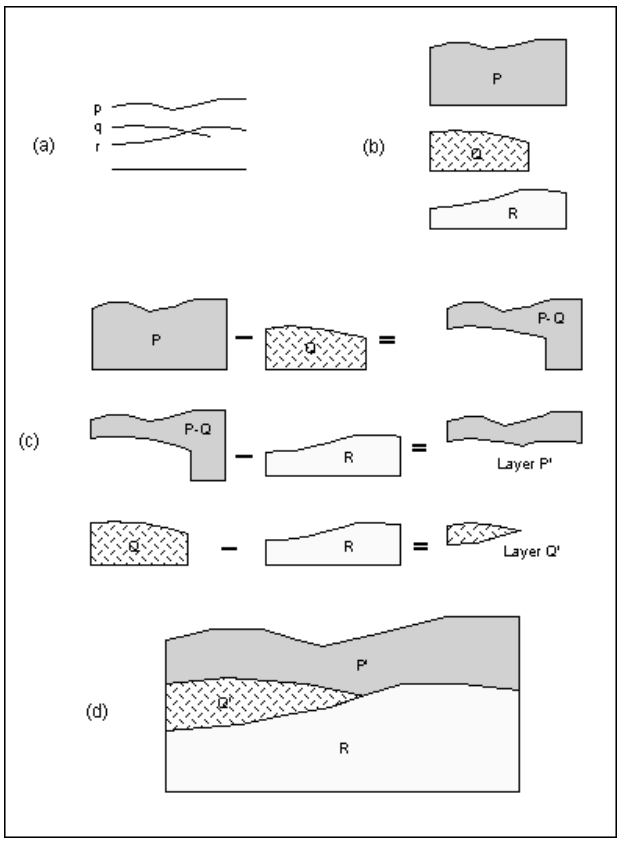
- **Horizons → Solids**
- **Horizons → 3D Mesh**
- **Fill Between TINs → 3D Mesh**

  See Creating a 3D Mesh

The preferred method for creating solids is the Horizons method mentioned above. The following commands are legacy operations that are less robust and not supported.

- **TINs -> Extruded Solid** – This command creates a new solid from each of the selected TINs by extruding each of the TINs up or down to an elevation specified by the user. Extruded TINs are useful in the construction of solid models of soil stratigraphy.
- **Fill Between TINs → Solid** – This command provides a quick way to create a solid bounded above and below by two or more selected TINs. The TIN defining the top of the boundary of the solid should be selected first. The remaining TIN(s) are then selected. All selected TINs are extruded down to an arbitrary elevation below that of all the selected TINs. GMS then performs a difference set operation.

TINs can be used to build three-dimensional solid models of the soil layers. The transformation from TINs to solids is accomplished using a TIN extrusion and set operation procedure illustrated in two dimensions in the following figure. A two-dimensional cross section of three TINs, labeled p, q, and r, is shown in part (a).



The TINs are converted into temporary solid primitives that represent approximations of the soil layers. The conversion is accomplished by projecting the outer boundary (perimeter) of each TIN down to a horizontal plane. This can be thought of as an extrusion process where a two-dimensional surface is extruded into a three-dimensional solid. A three-dimensional illustration of this process is shown in the figure below.

Boundaries are created around the perimeter of the solid and one large boundary is created at the base of the solid. The elevation of the horizontal plane is chosen so that the resulting solid is below the lowest point of interest. A series of two-dimensional cross sections of the primitive solids P, Q, and R formed by extruding the TINs in part (a) of the figure above is shown in part (b).

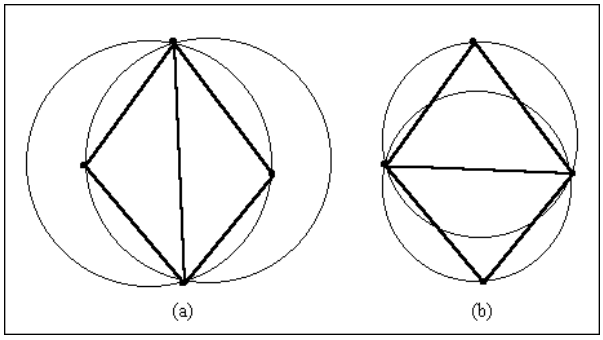
The final step of the modeling process consists of combining the primitive solids to form solid models of the soil layers. This is accomplished using set operations. Portions of the solids that overlap other solids are "trimmed" away and adjacent solids are forced to match precisely at the boundaries. This step of the modeling process is illustrated in part c. Primitive Q is subtracted from primitive P to produce the temporary solid P-Q. Primitive R is then subtracted from P-Q to produce the solid P'. The solid Q' is formed by subtracting primitive R from primitive Q. The primitive R does not intersect other solids and needs no trimming. Cross sections of the completed solid models of the soil layers are shown in part (d).

The combination extrusion/set operation process can be simplified in some cases. For example, within GMS it is possible to create solid P' directly by "filling" between TIN p and the two TINs q and r. GMS accomplishes this by combining the process described above for creating solid P' into a single operation. The user simply selects TINs p, q, and r and performs the ***Fill Between TINs → Solid*** command in the Build ***TINs*** menu of the TIN module.

The combination of TIN editing, TIN extrusion, and set operations represents a powerful and flexible tool that makes it possible to model complex stratigraphic relationships such as truncations, faults, embedded seams, and pinchout zones. Once the models are constructed, the volumes of the solids can be viewed using the Get Info command in the File menu. In addition, the models can be further modified using set operations to simulate complex excavations. Cross sections and fence diagrams can be constructed from the solid models at any location and at any orientation.

Triangulation

A TIN is constructed by triangulating a set of vertices. The vertices are connected with a series of edges to form a network of triangles. The resulting triangulation satisfies the Delauney criterion. The Delauney criterion ensures that no vertex lies within the interior of any of the circumcircles of the triangles in the network as shown below:

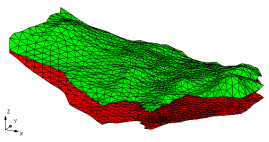


Two Adjacent Triangles Which (a) Violate and (b) Honor the Delauney Criterion.

The result of enforcing the Delauney criterion is that long thin triangles are avoided as much as possible.

The vertices associated with the active TIN can be triangulated using the ***Triangulate*** command from the ***TIN*** menu, or by right-clicking on the TIN in the Project Explorer and selecting the ***Triangulate*** command.

TIN Files

TIN Module

TINs
Creating a TIN
Editing a TIN
Converting TINS to Other Data Types
Building Solids and 3D Meshes with TINs
Map to MODFLOW with TINs
TIN Settings
More
TIN Settings
TIN Display Options
TIN Tool Palette
TIN Files

TIN files are used for storing Triangulated Irregular Networks. The TIN file format is shown below and a sample file is shown after. The TIN file format can be used to import a simple set of xyz coordinates since the triangle information (beginning with the TRI card) does not need to be present. If you have a file of xyz coordinates you only need to add the TIN, BEGT, and VERT nv cards to the top of the file and the ENDT card at the end.

```
TIN                 /* File type identifier */
BEGT                /* Beginning of TIN group */
TNAM name           /* Name of TIN */
TCOL id             /* TIN material id */
VERT nv             /* Beg. of vertices */
x1 y1 z1 lf1        /* Vertex coords. */
x2 y2 z2 lf2
.
.
.
xnv ynv znv lfnv
TRI nt              /* Beg. of triangles */
v11 v12 v13         /* Triangle vertices */
v21 v22 v23
.
.
.
vnt1 vnt2 vnt3
ENDT                /* End of TIN group */
```

Sample TIN File:

```
TIN
BEGT
TNAM Aspen
TCOL 255 255 255
VERT 408
0.0 3.1 7.8 0
5.3 8.7 4.0 1
.
.
2.4 4.4 9.0 1
TRI 408
5 1 4
4 1 2
.
.
4 2 3
ENDT
```

Cards used in the TIN file

Card Type	TIN
Card ID	3000
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	BEGT
Card ID	3000
Description	Marks the beginning of a group of cards describing a TIN. There should be a corresponding ENDT card at a latter point in the file. No fields.
Required	YES

Card Type	TNAM		
Description	Provides a name to be associated with the TIN.		
Required	NO		
Format	TNAM name		
Sample	`TNAM aspen`		
Field	**Variable**	**Value**	**Description**
1	name	str	The name of the TIN.

Card Type	TCOL		
Description	Defines a default color for the triangles of the TIN		
Required	NO		
Format	TCOL color_red color_green color_blue		
Sample	`TCOL 255 255 255`		
Field	**Variable**	**Value**	**Description**
1	color_red	0–255	The red color component of TIN triangles.
2	color_green	0–255	The green color component of TIN triangles.
3	color_blue	0–255	The blue color component of TIN triangles.

Card Type	MAT		
Description	Associates a material id with the TIN. This is typically the id of the material which is below the TIN.		
Required	NO		
Format	MAT id		
Sample	MAT 3		
Field	**Variable**	**Value**	**Description**
1	id	+	The material ID.

Card Type	VERT		
Description	Lists the vertices in the TIN		
Required	YES		
Format	VERT nv x_1 y_1 z_1 lf_1 x_2 y_2 z_2 lf_2 . . x_{nv} y_{nv} z_{nv} lf_{nv}		

Sample	`VERT 4`
	`0.0 3.1 7.8 0`
	`5.3 8.7 4.0 1`
	`2.4 4.4 9.0 1`
	`3.9 1.2 3.6 0`		

Field	**Variable**	**Value**	**Description**
1	nv	+	The number of vertices in the TIN
2–4	x,y,z	±	Coords. of vertex
5	lf	0,1	Locked / unlocked flag for vertex (optional). 0=unlocked, 1=locked. Repeat fields 2-5 nv times.

Card Type	**TRI**		
Description	Lists the triangles in the TIN		
Required	NO (a set of triangles can be generated from the vertices)		
Format	TRI nt v_{11} v_{12} v_{13} v_{21} v_{23} v_{23} . . v_{nt1} v_{nt2} v_{nt3}		

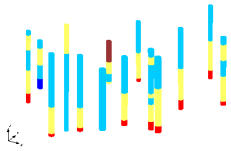
Sample	`TRI 4`
	`5 1 4`
	`4 1 2`
	`4 2 3`
	`5 4 3`		

Field	**Variable**	**Value**	**Description**
1	nt	+	The number of triangles in the TIN.
2–4	v1,v2,v3	+	Vertices of triangle listed in a counter-clockwise order. Repeat nt times.

Card Type	**ENDT**
Card ID	3000
Description	Marks the end of a group of cards describing a TIN. There should be a corresponding BEGT card at a previous point in the file. No fields.
Required	YES

5.2. Boreholes Module

Boreholes Module

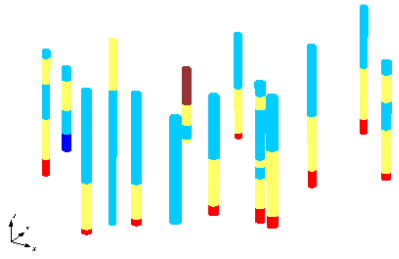
Boreholes Module

Boreholes Links
Creating and Editing Boreholes
Borehole Hydrogeologic Units
Converting Borehole Data
More
Borehole Cross Sections
Borehole Display Options
Borehole Tool Palette
Borehole Commands

Types of Borehole Data

A borehole can contain either stratigraphy data or sample data or both.

Stratigraphy

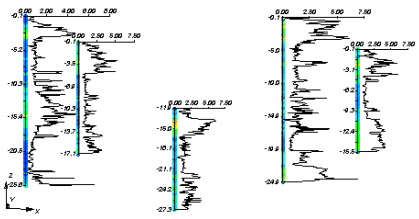
Stratigraphy data are used to represent soil layers that are encountered in a soil boring. The soil layers are represented using contacts and segments as shown below. A segment represents a soil layer and a contact is the interface between two segments.



Boreholes Representing Soil Stratigraphy With Contacts And Segments

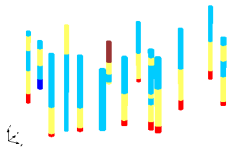
Sample Data

Sample data represent data obtained by continuous sampling along the length of the hole. Cone penetrometer data and down-hole geophysical data are examples of sample data. The figure below shows an example of sample data being displayed. Sample data are stored in datasets which can be manipulated in a similar fashion as other datasets in GMS.



Boreholes with Sample Data

Creating and Editing Boreholes

Boreholes Module

Boreholes Links
Creating and Editing Boreholes
Borehole Hydrogeologic Units
Converting Borehole Data
More
Borehole Cross Sections
Borehole Display Options
Borehole Tool Palette
Borehole Commands

Boreholes can be created by importing borehole data, importing sample data, or using the borehole tools to manually enter the boreholes.

To create a borehole data file, make a file with the borehole name, x, y, z, locations and a material ID. The z location will be the top of the soil layer. Once a borehole has been created and imported, it can be edited in the **Borehole Editor** or by using the borehole tools.

When you right click on a borehole in the Project Explorer you can copy the borehole by selecting the **Duplicate** command in the pop up menu. This will create a new borehole offset in x and y by 10% of the extents of the current data in GMS. This command is useful when there is a large gap between boreholes. A new borehole with similar stratigraphy to neighboring boreholes can be placed in the gap and the contacts can be positioned as desired. Adding an artificial borehole or a "pseudo-borehole" in the gap gives the user more control over the shape of the TINs and solids created from the boreholes.

Boreholes can be locked to prevent them from being edited. When the boreholes are locked, all graphical editing is disabled and a check appears on the menu. This prevents the boreholes, the borehole contacts, and the borehole segments from being inadvertently dragged with the mouse. Also, the Edit Window becomes disabled. The boreholes can still be edited using the Borehole Editor. The boreholes can be unlocked by selecting the ***Lock All Boreholes*** command again, and the check in the menu will disappear.

Auto Select

With a large number of boreholes, it may be tedious to individually select all the borehole contacts necessary for an operation. For this reason, the capability to automatically select multiple contacts is provided with the ***Auto Select*** command. One contact representing a prototype or example is first selected and the ***Auto Select*** command is chosen. The Auto-Select dialog appears showing a close-up of the selected contact and allows for:

- Matching of the material above, below, or both.
- Starting the search from the top or the bottom of the borehole.

Since only one contact per borehole is selected, the appropriate combination of the above options is important. Each borehole is searched from either the top or bottom of the hole until the first match is made. That contact is then added to the set of selected contacts.

The ***Auto Select*** command can also be used with the ***Select Segment*** tool in the Borehole Tool Palette to quickly select all segments matching a selected borehole segment. In this case, the segments are selected automatically and the Auto Select dialog does not appear.

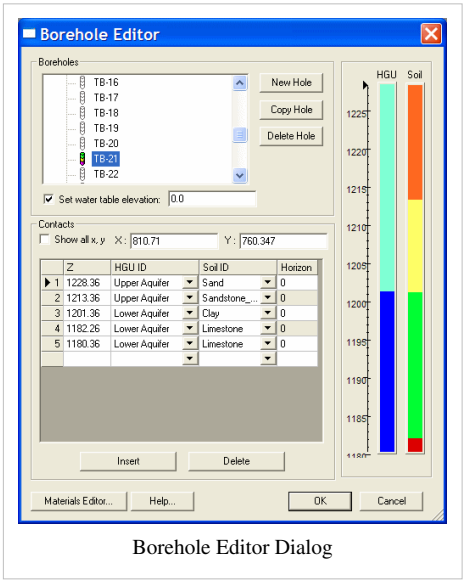
Borehole Editor

The Borehole Editor can be used to create new boreholes and edit existing boreholes. The existing boreholes are displayed in a tree window at the top of the dialog, with the currently selected borehole being highlighted. The currently selected borehole is drawn along the right side of the dialog. Both Hydrogeologic Units (HGU) and Soils are shown.

The name of the borehole can be changed by clicking on the borehole name in the text window and typing in a new name.

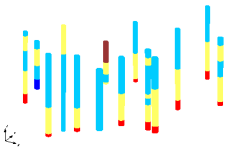
If the **Set water table elevation** toggle is on, a water table elevation can be entered. This can be used to display a water table symbol on each hole.

The borehole's contacts are listed in the spreadsheet in the middle of the dialog. Contacts can be deleted and new contacts can be inserted above the currently selected contact using the buttons just below the spreadsheet. The material below the contact is specified by selecting the material name.



Borehole Editor Dialog

Borehole Display Options

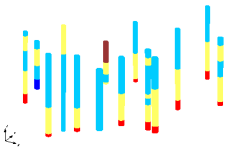
Boreholes Module

Boreholes Links
Creating and Editing Boreholes
Borehole Hydrogeologic Units
Converting Borehole Data
More
Borehole Cross Sections
Borehole Display Options
Borehole Tool Palette
Borehole Commands

The properties of all borehole data that GMS displays on the screen can be controlled through the Borehole tab of the Display Options dialog. This dialog is opened by right clicking on the Borehole Data entry in the Project Explorer and selecting the ***Display Options*** command. It can also be accessed from the from the *Display* menu or the Display Options macro. The following table describes the display options available for the Borehole module.

Display Option	Description
Borehole edges	This option controls the display of the lines that show the outline of the boreholes. The color of the borehole edges can be adjusted according to the following options: 1. Auto – draws the material color if faces are not displayed. Uses black or white if the faces are displayed 2. Specified – used the color specified next to the borehole edges 3. Material – displays the material color of the borehole segment
Borehole faces	If this option is on then the borehole segments are displayed are filled polygons.
Diameter	This value determines the display size of the boreholes in the graphics window in world length coordinates.
Num. slices	This edit field determines the number of slices to display the borehole. The default is 6 making the boreholes display as hexagons.
Hole names	If the Hole names box is checked, the name of each hole is displayed at the top of the hole.
Water table	If the Water table box is checked, an icon representing the water table is displayed at the water table elevation of each borehole.
Horizon IDs	The horizon IDs toggle controls the display of the horizon id next to each borehole contact.
Cross sections edges	The horizon ids toggle controls the display of the horizon id next to each borehole contact. 3 options are available for the cross section edges: 1. Auto – draws the material color if faces are not displayed. Uses black or white if the faces are displayed 2. Specified – used the color specified next to the cross section edges 3. Material – displays the material color of the stratigraphic unit in the cross section
Cross section faces	If this option is on then the borehole segments are displayed as filled polygons.
Cross section names	If this option is on then the borehole names are displayed above the boreholes.

Material display	This radio group determines the display color of the boreholes. The borehole segments can be colored either by the Soil or HGU assigned.
Cross Section Highlighting for Horizon Coverages	Toggle display of the lines showing the part of the cross sections where the material with the horizon ID of the active horizon coverage exists. This highlighting only appears when you are in plan view and the active coverage is a horizon coverage.
Points	If the Points box is checked, every sample data point is displayed. If the Use color ramp box is checked, the points are colored according to the current dataset and the current color ramp settings.
Lines	If the Lines box is checked, the sample points are connected by a series of line segments. If the Use color ramp box is checked, the line segments are colored according to the current dataset and the current color ramp settings.
Data plots	If the Data plots box is checked, a plot of the current data set is drawn next to each borehole with sample data. The width (horizontal length) can be adjusted and the options associated with the plot scale, plot axes, etc., can be accessed by selecting the Plot Options button.
Data range	By default, the minimum color on the color ramp is associated with the minimum data set value and the maximum color is associated with the maximum data set value. The ramp of colors can be confined to a smaller interval defined by the Maximum and Minimum values. This forces all of the color gradation to be concentrated in a particular range of interest.

Borehole Tool Palette

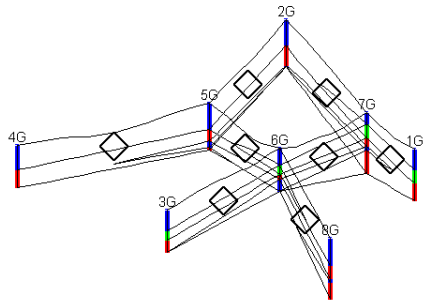
Boreholes Module

Boreholes Links
Creating and Editing Boreholes
Borehole Hydrogeologic Units
Converting Borehole Data
More
Borehole Cross Sections
Borehole Display Options
Borehole Tool Palette
Borehole Commands

The following tools are available in the dynamic portion of the Tool Palette whenever the Borehole Module is activated. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the borehole tool palette.

Tool	Tool Name	Description
	Select Borehole	The Select Borehole tool is used to select entire boreholes. Information about the selected borehole can be obtained by using the Get Info command from the File menu. Selected boreholes can be deleted, or dragged with the mouse. In plan view, the borehole can be dragged anywhere in the XY plane. In other views, the borehole can only be dragged up and down along the Z axis unless the Control key is held down, in which case the borehole can be dragged anywhere in the viewing plane. The coordinates of the top of the borehole can be edited in the Edit Window. The name associated with a selected borehole can be edited by double-clicking on the borehole or by selecting the Attributes command from the Edit menu while the borehole is selected.
	Select Segment	The Select Segment tool is used to select the region between two contacts. Information about the selected segment can be obtained by using the Get Info command from the File menu. The selected segment can be deleted unless it is the only segment on the borehole. In plan view, the segment can be dragged anywhere in the XY plane with the mouse. In other views, the selected segment can only be dragged up and down along the Z axis, unless the Control key is held down, in which case the segment can be dragged anywhere in the viewing plane. The coordinates of the top contact on the segment can be edited in the Edit Window . The material associated with the segment can be changed by double-clicking on the segment or by using the Attributes command in the Edit menu. Several segments with the same material type can be selected automatically by using the Auto Select command or they can be selected sequentially while holding down the Shift key.
	Select Contact	The Select Contact tool is used to select the interfaces between soil layers. Selected contacts can be deleted as long as there are at least two contacts remaining on the borehole after deletion. In plan view, the selected contact can be dragged anywhere in the XY plane with the mouse. In other views, the selected contact can only be dragged up and down along the Z axis, unless the Control key is held down, in which case the contact can be dragged anywhere in the viewing plane. The coordinates of the contact can be edited in the Edit Window. Multiple contacts can be selected sequentially by holding down the Shift key, or they can be selected automatically using the Auto Select command. Selected contacts can be used to create TINs. A horizon id can be assigned to a selected contact(s) by selecting the Properties command in the Edit menu.
	Select Cross Section	The Select Cross Section tool selects existing cross sections by clicking on the selection icon in the GMS graphics window when this tool is active. See the figure below.

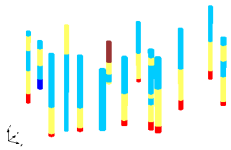
	Create Borehole	The Create Borehole tool can be used to create a new borehole at the location clicked on by the mouse. The user is first prompted for the missing coordinate (i.e., in plan view, the z coordinate is asked for). Boreholes can not be created in oblique view. The borehole is given a default name of "New Borehole" and three segments which are ten units long by default. A newly created borehole can be edited using the other tools in the Tool Palette or the Borehole Editor.
	Create Contact	The Create Contact tool can be used to create a new contact on an existing borehole by clicking on the borehole at the location where the new contact is to be located. The user is then prompted for the material associated with the contact (the material for the segment below the contact).
	Create Cross Section	The Create Cross Section tool creates user defined cross sections between existing boreholes. To create a single cross-section, the user clicks on the first hole and then double-clicks on the second hole. Multiple panels of a cross section can be created at once (i.e., a fence diagram) by single-clicking on sequence of boreholes and double-clicking on the last borehole.

The figure below shows a set of borehole cross sections. The cross section selection icon is the black diamond near the center of each of the cross sections. A cross section can be edited by selecting its corresponding icon and selecting the ***Cross Section Editor*** command from the *Borehole* menu or by double-clicking on the selection icon.



Set of borehole cross sections

Borehole Hydrogeologic Units

Boreholes Module

Boreholes Links
Creating and Editing Boreholes
Borehole Hydrogeologic Units
Converting Borehole Data
More
Borehole Cross Sections
Borehole Display Options
Borehole Tool Palette
Borehole Commands

Hydrogeologic units (HGUs) can be defined on boreholes. HGUs are typically a simplified representation of the soil layers from the borehole field data. For example, the borehole log may include several types of sand ("brown sand", "gray silty sand", "clean sand"), but for modeling purposes, you may want to treat these all as one material, "sand". Now you can show both the original soils and the simplified HGUs on the boreholes.

Importing Borehole Data

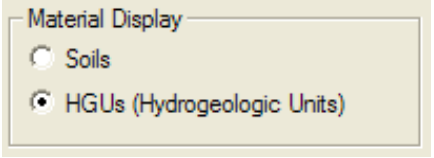
When importing borehole data, both an HGU and Soil ID column can be specified in the input. Files containing only a single set of materials can be imported to either field and the other field can be populated using the conversion tools described below.

Creation

The HGU and soil IDs can be edited using the Borehole Editor.

Display

The boreholes can be displayed in the main graphics window using either the HGU IDs or the soil IDs. The ID used for display can be selected in the Display Options dialog.



Borehole Display Options

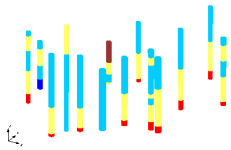
Soils → HGUs, HGUs → Soils

A set of soil IDs can be converted to a set of HGU IDs using the ***Soils→HGUs*** command in the Borehole menu. Likewise, a set of HGU IDs can be converted to a set of soil IDs using the ***HGUs→Soils*** command.

Building Cross-Sections, Solids

When building cross-sections or solid models using boreholes, the HGU IDs are used by GMS. The soil IDs are used purely for visualization or for setting up the HGU IDs.

Converting Borehole Data

Boreholes Module

Boreholes Links
Creating and Editing Boreholes
Borehole Hydrogeologic Units
Converting Borehole Data
More
Borehole Cross Sections
Borehole Display Options
Borehole Tool Palette
Borehole Commands

Borehole data can be converted to other types of objects with in GMS such as 2D Scatter Points, TINs, 3D Meshes. Borehole data is converted by using the following commands in the Boreholes menu:

- Horizons to Solids
- Horizons to HUF
- Horizons to 3D Mesh
- Contacts to TIN

  The ***Contacts → TIN*** command is used to create a TIN surface from a set of selected contacts.

- Contacts to 2D Scatter Points

  A set of selected contacts can be converted to a 2D scatter point set using the ***Contacts → 2D Scatter Points*** command.

- Sample Data to 3D Scatter Points

  The ***Sample Data → 3D Scatter Points*** command brings up the *Sample Data → Scatter Points* dialog that is used to create a 3D scatter point set from sample data.

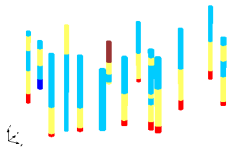
- Sample Data to Stratigraphy
- Watertable to 2D Scatter Points

  The water table coordinates for a set of boreholes can be converted to a 2D scatter point set using the ***Water Table → 2D Scatter Points*** command.

- Add Contacts to TIN

  The ***Add Contacts to TIN*** command is used to enter a point from a contact into the active TIN. The contact(s) are first selected and the command is then chosen from the *Borehole* menu. Typically all contacts which should be part of a TIN are selected before generating the TIN, but sometimes one is inadvertently left out, or more boreholes are added later.

Borehole Cross Sections

Boreholes Module

Boreholes Links
Creating and Editing Boreholes
Borehole Hydrogeologic Units
Converting Borehole Data
More
Borehole Cross Sections
Borehole Display Options
Borehole Tool Palette
Borehole Commands

A borehole cross section is a set of polylines and polygons that define the stratigraphy between two boreholes. A borehole cross section can be created manually or automatically.

Creation

Automatic creation

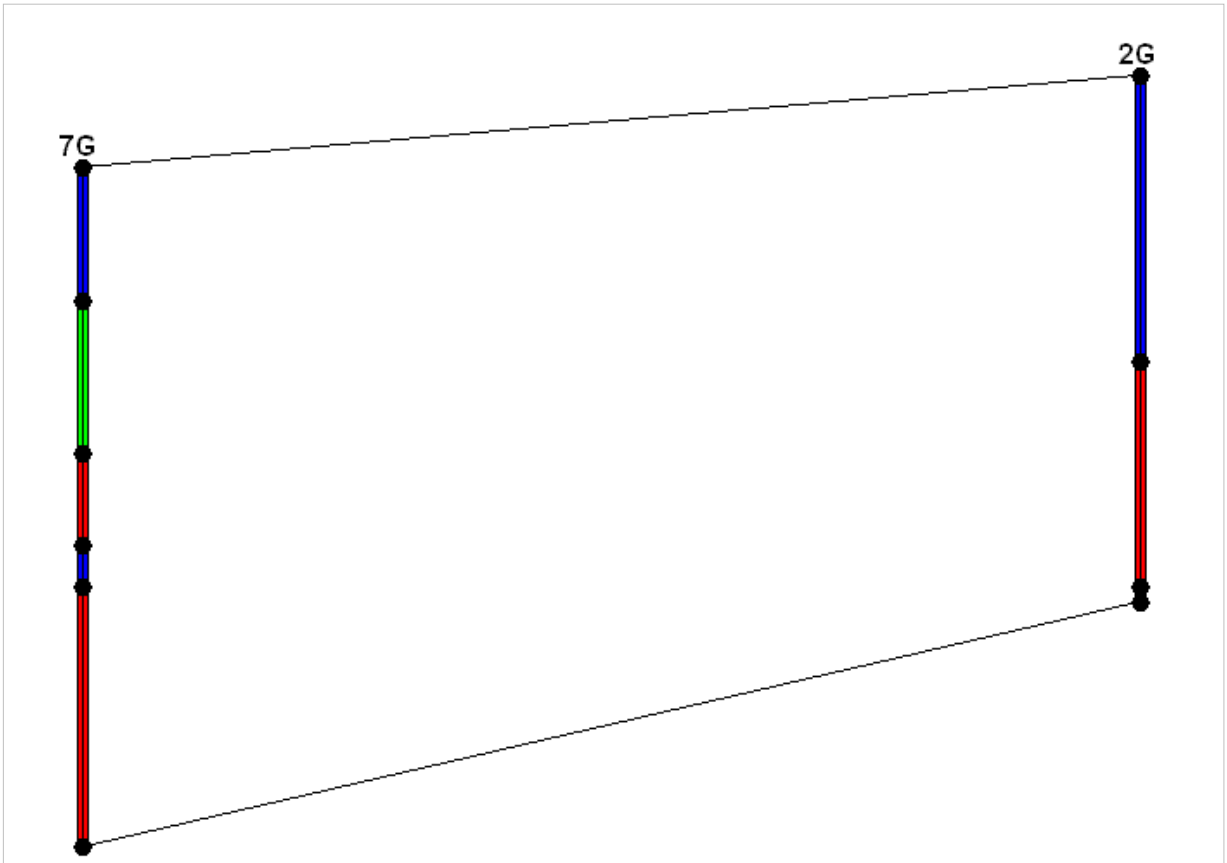
Cross sections can be created automatically using the ***Boreholes\Auto-Create Blank Cross Sections*** menu command. This uses a triangulation process to determine the most likely connections between boreholes. The top and/or bottom arcs of the new cross sections can be warped to match the elevation of TIN surfaces using the *Snap Cross Sections to TIN* dialog which appears when the ***Boreholes\Auto-Create Blank Cross Sections*** command is executed. Snapping the tops and bottoms of cross sections to TIN surfaces can also be done at any time via the ***Boreholes\Advanced\Snap Cross Sections to TIN*** menu command. Keep in mind, however, that warping the top and bottom of a cross section may interfere with the internal polygons that are defined in the cross section, so snapping to a TIN is best done before filling in the cross sections.

Manual creation

Cross sections can be created manually by using the Create Cross Section tool and clicking on boreholes.

Editing

When a borehole cross section is first created, it is made up of a set of default lines. The figure below shows a default cross section. Notice that "arcs" (polylines) have been created defining the top and bottom of the cross section and an "arc" (polyline) has been created for each segment in the boreholes.



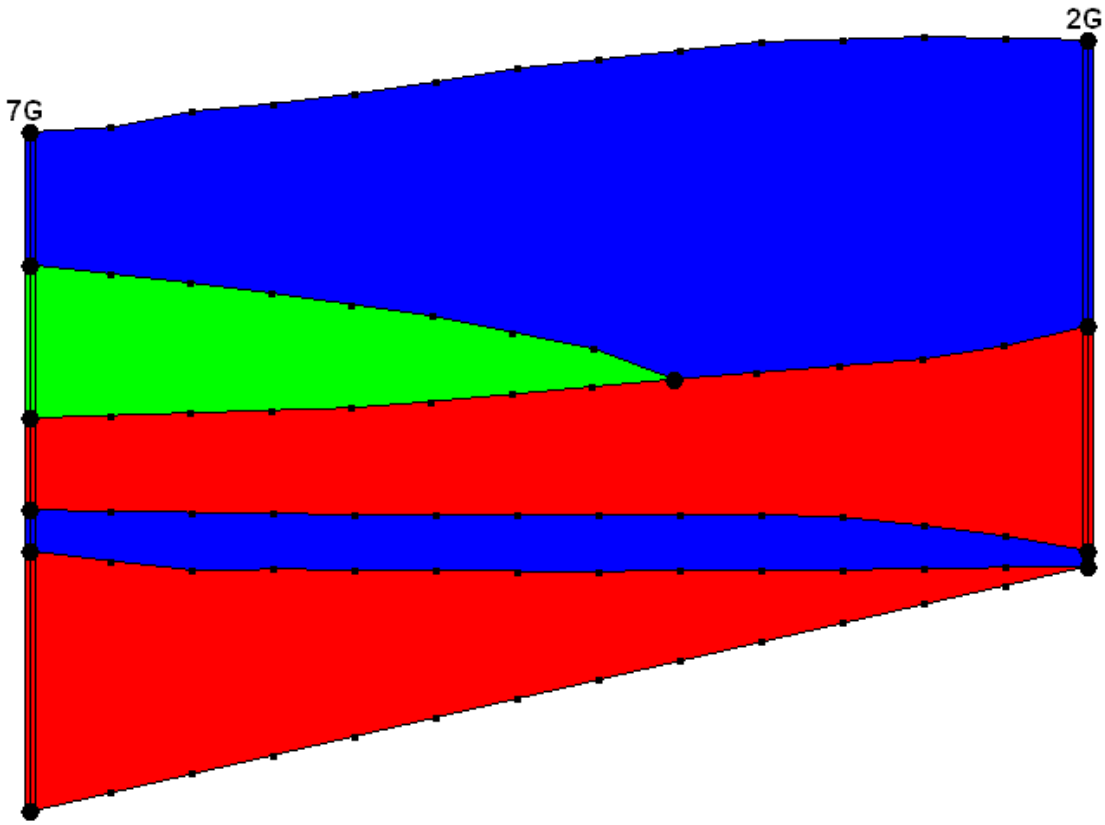
Default cross section

Automatic editing

The ***Boreholes\Auto-Fill Blank Cross Sections*** menu command can be used to automatically fill in all existing blank cross sections. The command can use either the horizon IDs (preferred) or materials information on the boreholes.

Manual editing

To edit a cross section manually, the user must use the Select Cross Section tool and select a cross section. The *Cross Section Editor* dialog can be then opened by selecting the ***Cross Section Editor*** command from the *Borehole* menu while the user has either a single or a series of cross sections selected. The user can also launch the Cross Section Editor dialog by double-clicking on a single cross section. The next figure shows a finished cross section.



Complete cross section

Borehole Cross Section Editor

The Cross Section Editor can be used to manually construct and view cross sections between boreholes. The Cross Section Editor in GMS 6.5 has been updated to allow for the display and editing of multiple borehole cross sections. The Cross Section Editor has a set of tools and toggles used to create, edit, and view the cross sections. The tools used to create the nodes, polylines, and polygons are similar to the tools available in the Map module used to create feature objects (points, arcs, and polygons). The following tables describe the tools and display options available in the Cross Section Editor.

Tools

Tool	Tool Name	Description
	Select Tool	Generic selection tool that selects nodes, vertices, arcs, and polygons
	Select Point/Node	Selection tool that will only select points or nodes
	Select Vertex	Selection tool that will only select vertices
	Select Arc	Selection tool that will only select arcs
	Select Polygon	Selection tool that will only select polygons
	Create Vertex	Creates new vertices along arcs within the cross section
	Create Arc	Creates arcs between two nodes or vertices within the cross section
	Pan	Pans in the viewing area of the Graphics Window
	Zoom	Magnifies or shrinks the current viewing area

	Frame All Cross Sections	Frames to the extents of all the cross sections
	Frame Current Cross Section	Frames to the extents of the current cross section
	Z-Magnification	Adjusts the Z-Magnification factor to increase or decrease the graphical display along the Y (real world Z) axis, making more or less room at the top and bottom of the screen while maintaining the boreholes and cross sections in the middle of the screen
	Plot Options	Adjusts the axes plot options, including: title, background color, font, font color, grid display, axes titles, and axes display
	Delete	Deletes the currently selected vertices, nodes, arcs, or polygons
	Left	Activates the cross section to the left of the current cross section as the current cross section
	Right	Activates the cross section to the right of the current cross section as the current cross section
	Print	Prints the current Graphics Window
	Auto-Match Cross Section	Creates a set of straight arcs connecting matching contacts on adjacent boreholes based on the user's selection to use Horizons or Materials
	Build Cross Section Polygons	Deletes all current polygons, builds new polygons using all of the arcs, and checks to see if every polygon built is valid. A polygon is valid only if it contains either one arc representing a borehole region or two arcs representing two matching borehole regions on two holes. Thus, every valid polygon can be assigned one and only one material type. If every polygon built is valid, the Color Fill toggle will be automatically turned on and all polygons built will be filled with the color representing the material they are assigned. Otherwise, a dialog saying "Invalid polygons present" will pop up and all polygons built will be deleted.
	Delete All	Deletes all of the vertices, nodes, arcs, and polygons in the current cross section

Display Options

Display Toggle	Description
Nodes	Controls the display of the nodes in the graphics window
Vertices	Controls the display of the vertices in the graphics window
Arcs	Controls the display of the arcs in the graphics window
Boreholes	Controls the display of the arcs in the graphics window
Poly fill	Controls the display of the polygons in the graphics window
Mirror view	Reverses the order in which the cross sections are displayed
Mark inactive	Dims the inactive cross sections
Display axes	Controls the display of the plot axes in the graphics window

5.2.1. Horizons

Horizons

Horizons
Links
Horizons
Horizons to Solids
Horizons to 3D Mesh
Horizons to HUF
Horizons Wizard
Including Borehole Cross Sections
Raster Catalog
Horizons Conceptual Model
Horizons Applications

The term “horizon” refers to the top of each stratigraphic unit that will be represented in a corresponding Solid, HUF unit or Material Layer. Horizons are numbered consecutively in the order that the strata are “deposited” (from the bottom up). Horizons can be assigned to both Boreholes and TINs. Beginning with version 9.0, raster catalogs can also be used to define horizons.

Once horizons have been assigned to boreholes and/or TINs, the Horizons Wizard can be used to create solids, 3D mesh, or HUF data.

Assigning Horizons to Boreholes

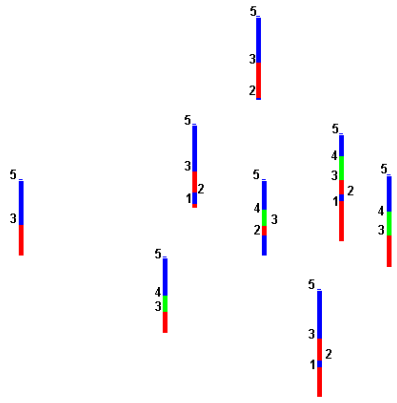
Horizons are defined at borehole contacts. Each contact that the user wishes to include in the construction of the solid must have a horizon ID. If the user wishes to ignore a contact, this can be done by setting the horizon ID to zero. Horizons are numbered in the order that the strata are “deposited” (from the bottom up). Gaps can exist in the horizons. For example, horizons can be assigned using 1, 2, 3, ect..., or the user could assign horizons using 10, 20, 30, etc...

Automatic Assignment

To have GMS automatically assign horizon IDs to boreholes, you can use the ***Boreholes | Auto-Assign Horizons*** menu command. Depending on the number and complexity of your boreholes, this command may take a considerable amount of time.

Manual Assignment

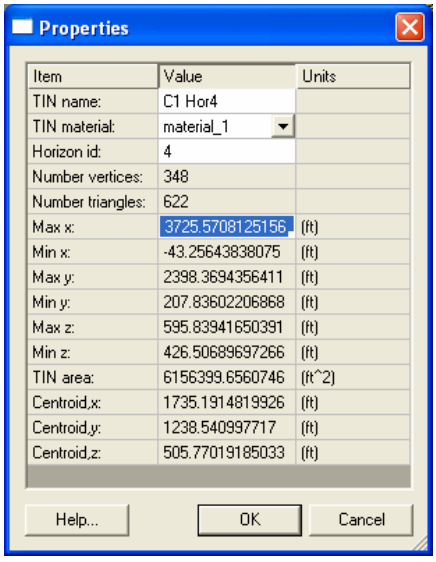
Horizons are defined at borehole contacts (interface between different materials on a borehole log) by double clicking on a contact with the ***Select Contact*** tool. The ***Boreholes | Auto Select*** command can be helpful in assigning horizons to a large group of boreholes.



Horizons assigned to contacts on boreholes

Assigning Horizons to TINs

A TIN Horizon is assigned in the TIN properties dialog. This dialog can be accessed by right-clicking on a TIN in the Project Explorer and selecting the properties command. Each TIN can be assigned one Horizon ID.



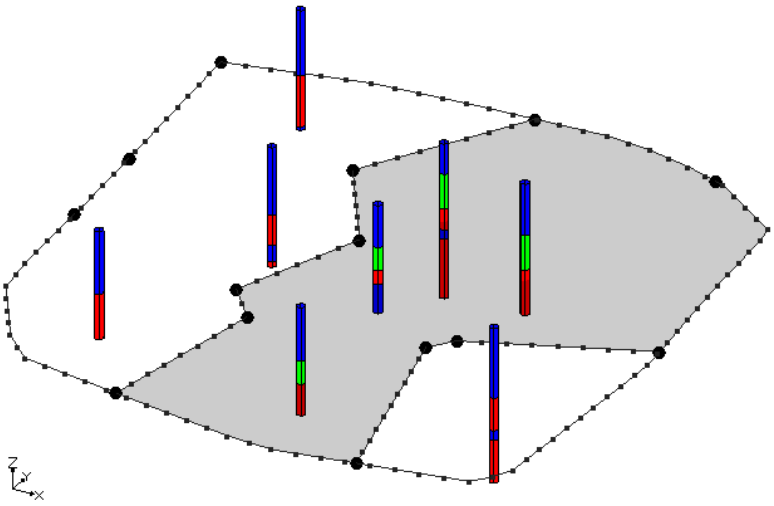
Horizons assigned to contacts in TIN Properties Dialog

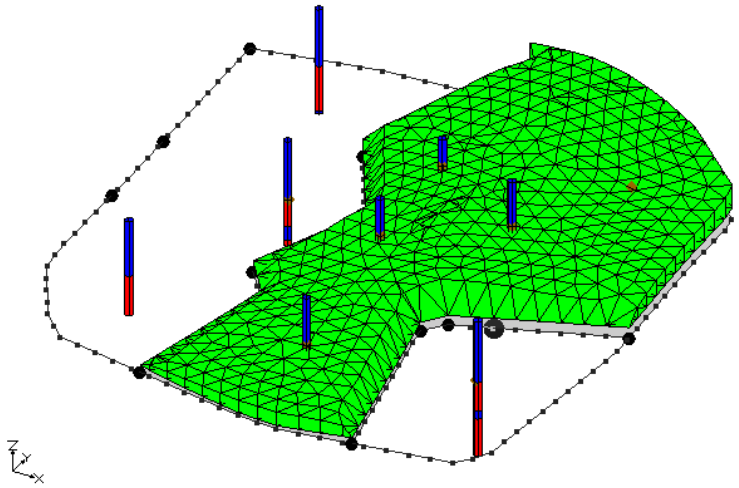
Each TIN that the user wishes to include in the horizons algorithm must have a horizon ID. If the user wishes to ignore a TIN, this can be done by setting the horizon ID to zero.

Horizon Conceptual Model

Horizons
Links
Horizons
Horizons to Solids
Horizons to 3D Mesh
Horizons to HUF
Horizons Wizard
Including Borehole Cross Sections
Raster Catalog
Horizons Conceptual Model
Horizons Applications

If the user wishes to explicitly control the areal extent of a solid created from horizons, this can be done using horizon coverages. GMS has a conceptual model type for horizons. Coverages that are inside of a horizons conceptual model can be associated with a horizon ID. The polygons in a horizon coverage determine the areal extent of the solid for the associated horizon. The first figure below shows a horizon coverage for the green material. The solid associated with this horizon will not extend beyond the boundary of the polygon. The second figure shows the solid resulting from the boreholes and the horizon coverage. When the Horizons→Solids command is executed the user may include a Horizons conceptual model as part of the input to the command.





The Horizons algorithm is used to create either solids, HUF units, or 3D Meshes from Borehole and TIN data. How to use the three Horizon commands are explained in more detail below:

- Horizons to Solids
- Horizons to 3D Mesh
- Horizons to HUF

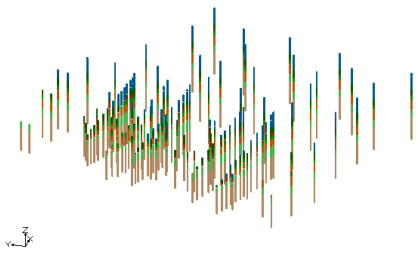
Horizons Applications

Horizons
Links
Horizons
Horizons to Solids
Horizons to 3D Mesh
Horizons to HUF
Horizons Wizard
Including Borehole Cross Sections
Raster Catalog
Horizons Conceptual Model
Horizons Applications

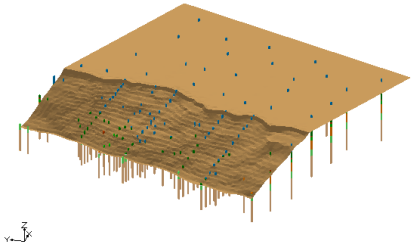
The horizons method has been applied at a variety of sites to construct solid models of the subsurface. This page highlights example applications of the Horizons Method.

Modeling a Slope Failure

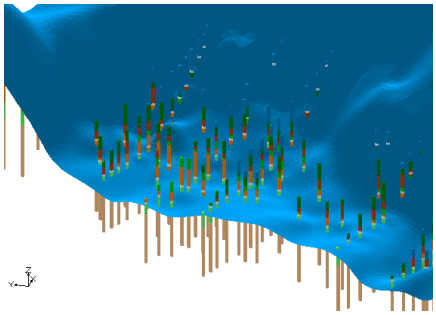
In this example a combination of boreholes, user defined cross sections, and TINs were used to create solids at a site with a slope failure.



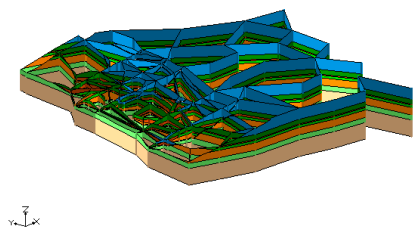
Borehole data



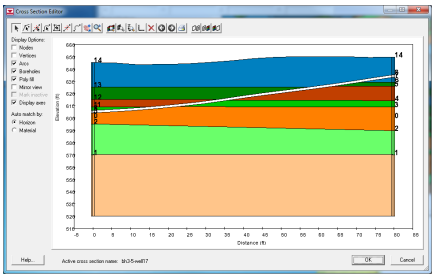
Surface elevations



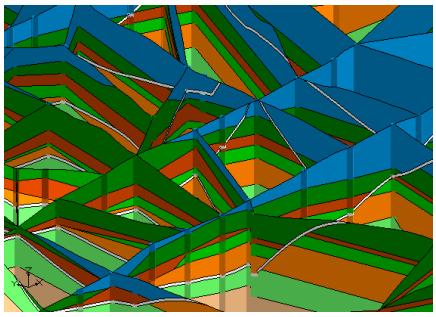
Failure surface



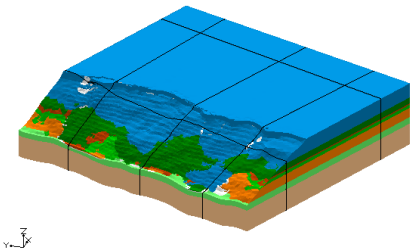
User defined cross sections



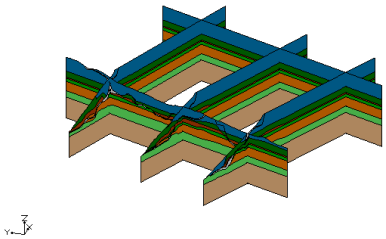
One user cross section



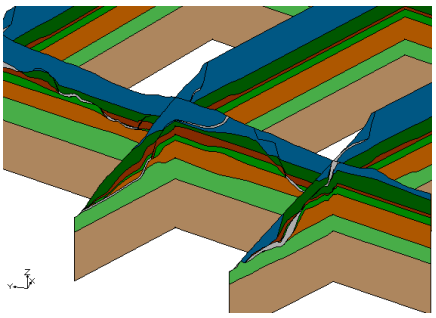
Zoomed in on cross sections



Solids created from user data



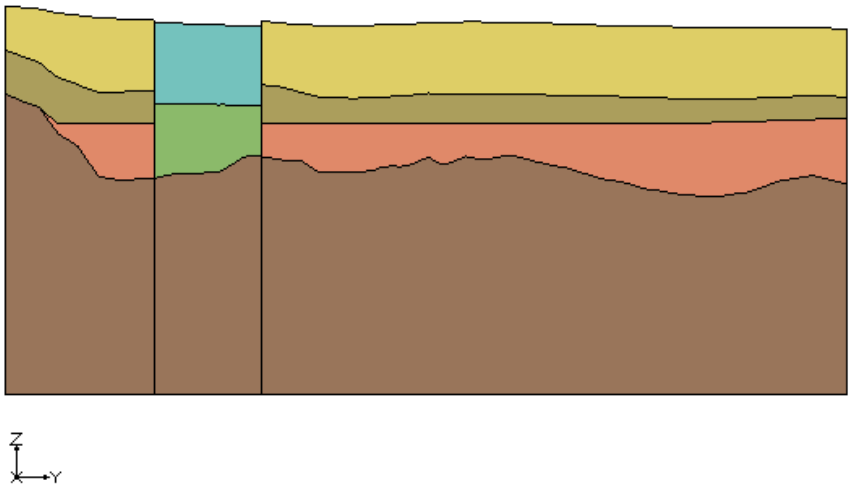
Cross sections from solids



Zoomed in on cross sections

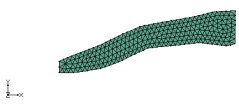
Vertical Boundary Between Solids

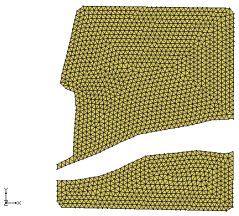
In this application the user wanted to create a set of solids where there would be distinct materials below a river bed compared to the other materials in the study area. The following cross section shows what the user wanted to create.



Primary TINs

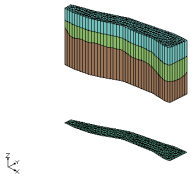
To create solids that would match this cross section, the user created 2 different primary tins and executed the Horizons->Solids command for each primary TIN. The first TIN covered the area of the river and the second TIN covered the remainder of the study area as show in the images below.

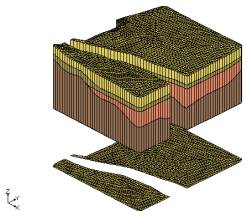




Solids

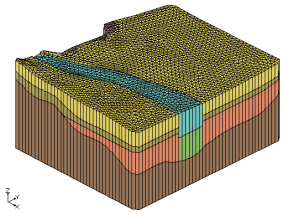
The user had TINs that defined the top elevations of the horizon surfaces. There were 3 TINs used in the area around the river and there were 4 TINs used the remainder of the study area. Solids were created for the river area using the first TIN as the primary TIN. Solids were also created in remainder of the study area by using the second TIN as the primary TIN. Notice the the bottom most material matches in both sets of solids. This is because the same TIN with that horizon was used when creating both sets of solids.

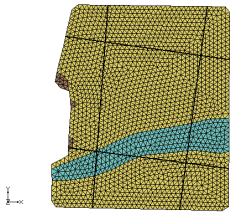


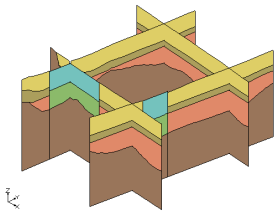


Cross Sections from Solids

These images show the solids together and cross section cut through the solids. Again notice how the bottom most material matches across both sets of solids.







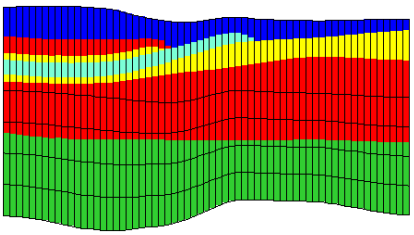
Horizons to HUF

Horizons
Links
Horizons
Horizons to Solids
Horizons to 3D Mesh
Horizons to HUF
Horizons Wizard
Including Borehole Cross Sections
Raster Catalog
Horizons Conceptual Model
Horizons Applications

The following steps illustrate how to use the Horizons method to create HUF data

1. **Create/Import Inputs –** There are two main types of inputs for the horizons method:
   1. Boreholes – Boreholes can be created by importing borehole data by using the Text Import Wizard, importing sample data after boreholes already exist, or using the borehole tools to manually enter the boreholes. Once a borehole has been created it can be edited in the Borehole Editor or by using the borehole tools. Also an existing borehole can be copied. Boreholes can be locked to prevent them from being edited.
   2. TINs – TINs can be created 3 different ways in GMS: manually entering the vertex locations and triangulating, converting a different GMS data type to a TIN, and copying a currently existing TIN. (See Creating TINs)
   3. Raster Catalog - a set of rasters defining the top of each horizon (available beginning in version 9.0).
2. **Assign Horizon IDs –** The term “horizon” refers to the top of each stratigraphic unit that will be represented in a corresponding Solid, HUF unit or Material Layer. Horizons are numbered consecutively in the order that the strata are “deposited” (from the bottom up). Horizons can be assigned to both Boreholes and TINs. (See GMS:Horizons)
3. **Create 3D Grid/MODFLOW model –** A 3D Grid and MODFLOW model need to be first created to use the Horizon -> HUF command. The flow package for the MODFLOW model must also be set to use the HUF package. Before building a MODFLOW simulation, a 3D grid must be created which covers the area to be modeled. A grid can be created by selecting the ***Create Grid*** command in the *Grid* menu. A suite of tools and commands for editing grids (inserting rows, changing column widths, etc.) are also provided in the 3D Grid Module. If the conceptual model approach is used to construct a MODFLOW model, the grid can be automatically constructed from the conceptual model data using the ***Grid Frame*** and the ***Map → 3D Grid*** command in the *Feature Objects* menu. The grid can be automatically refined around wells and cells outside the model domain can be inactivated.
4. **Setup additional optional inputs –** Two additional options exist to help constrain and provide user intervention in the Horizon modeling process. The two options are to create borehole cross sections or a horizon conceptual model.
   1. Including Borehole Cross Sections
   2. Horizon Conceptual Model
5. **Run the Horizons Wizard –** Select the ***Horizons→3D Mesh*** command in the *Borehole* or *TINs* menu.

Horizon → HUF Algorithm



Example of Horizons to HUF

When the Horizon command is executed the horizons specified on the borehole contacts or TIN nodes are converted to a set of scatter points with one data set for each horizon. The scatter points are then used to interpolate a surface for each horizon. Starting with the lowest numbered horizon, the surface is extruded down to create a HUF layer The surface corresponding to the next horizon is then extruded down to fill in the space between that surface and the previous surface. This process is repeated for each surface. At each step, HUF layer is created for the current horizon and all previously layers are subtracted from that layer, resulting in an incremental buildup of the stratigraphy from the bottom to the top. In conclusion the HUF Package elevation and thickness arrays are generated from the horizon data.

Horizons Wizard

Horizons
Links
Horizons
Horizons to Solids
Horizons to 3D Mesh
Horizons to HUF
Horizons Wizard
Including Borehole Cross Sections
Raster Catalog
Horizons Conceptual Model
Horizons Applications

The Horizons Wizard is used to create solids, a 3D mesh, or HUF layers from horizon data. The wizard is started via the *Horizons → Solids*, *Horizons → 3D Mesh*, and *Horizons → HUF* commands. These commands are in the TIN and Boreholes menus.

Step 1

The first step is to define the inputs to be used, which can include boreholes, TINs, and a Horizon Conceptual Model.

Beginning with GMS 9.0, you can also include a raster catalog as input.

Step 2

The second step is to define the top and bottom of the solid, mesh, or HUF layers. When creating HUF data, you may also edit the grid elevations.

Step 3

The third step is to define the interpolation method to be used, as well as options specific to creating solids, a mesh, or HUF package.

Beginning with GMS 7.0, when creating solids, the user can choose the option *Preserve projection TIN datasets*. This option will create a new TIN that will have a dataset for each horizon. This is often useful so that the user can see the result of the interpolation process for each Horizon. The user can then edit the TIN by hand and include the TIN when executing the Horizons→Solids command.

Horizons to Solids

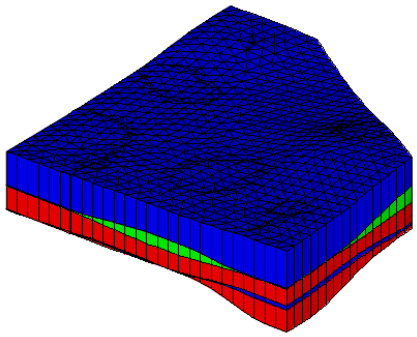
Horizons
Links
Horizons
Horizons to Solids
Horizons to 3D Mesh
Horizons to HUF
Horizons Wizard
Including Borehole Cross Sections
Raster Catalog
Horizons Conceptual Model
Horizons Applications

The following steps illustrate how to use the Horizons method to create solid stratigraphy.

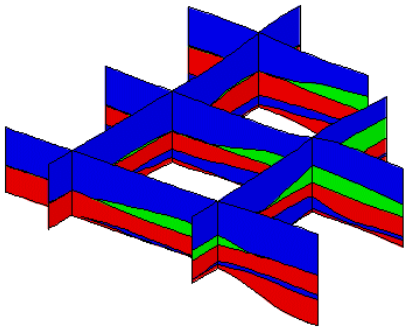
1. **Create/Import Inputs –** There are two main types of inputs for the horizons method:
   1. Boreholes – Boreholes can be created by importing borehole data by using the Text Import Wizard, importing sample data after boreholes already exist, or using the borehole tools to manually enter the boreholes. Once a borehole has been created it can be edited in the Borehole Editor or by using the borehole tools. Also an existing borehole can be copied. Boreholes can be locked to prevent them from being edited.
   2. TINs – TINs can be created 3 different ways in GMS: manually entering the vertex locations and triangulating, converting a different GMS data type to a TIN, and copying a currently existing TIN. (See Creating TINs)
   3. Raster Catalog – a set of rasters defining the top of each horizon (available beginning in version 9.0).
2. **Assign Horizon IDs –** The term "horizon" refers to the top of each stratigraphic unit that will be represented in a corresponding Solid, HUF unit or Material Layer. Horizons are numbered consecutively in the order that the strata are "deposited" (from the bottom up). (See Horizons)
3. **Create Primary TIN –** A TIN must be created or imported into GMS to be used as the Primary TIN for the Horizons method. The primary TIN defines the boundary of the solids that will be generated. Also, the density of the triangles in the primary TIN controls the density of the triangles in the solids that are created. (See Creating TINs)
4. **Setup additional optional inputs –** Two additional options exist to help constrain and provide user intervention in the Horizon modeling process. The two options are to create borehole cross sections or a horizon conceptual model.
   1. Including Borehole Cross Sections
   2. Horizon Conceptual Model
5. **Run the Horizons Wizard –** Select the ***Horizons→Solids*** command in the *Borehole* or *TINs* menu.

Horizon → Solid Algorithm

When the Horizon command is executed the horizons specified on the borehole contacts or TIN nodes are converted to a set of scatter points with one dataset for each horizon. The scatter points are then used to interpolate a surface for each horizon. Starting with the lowest numbered horizon, the surface is extruded down to create a solid. The surface corresponding to the next horizon is then extruded down to fill in the space between that surface and the previous surface. This process is repeated for each surface. At each step, a solid is created for the current horizon and all previously defined solids are subtracted from that solid, resulting in an incremental buildup of the stratigraphy from the bottom to the top. The entire process is simpler, more intuitive, and more robust than the old set operations approach.



Solids created from the horizons method



Cross sections from the solid created via the horizons method

Horizons to 3D Mesh

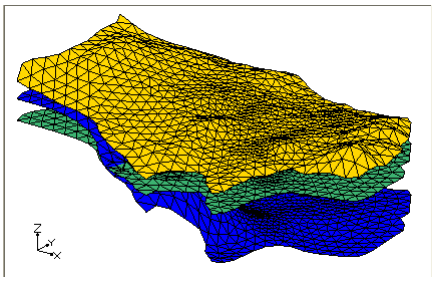
Horizons
Links
Horizons
Horizons to Solids
Horizons to 3D Mesh
Horizons to HUF
Horizons Wizard
Including Borehole Cross Sections
Raster Catalog
Horizons Conceptual Model
Horizons Applications

The following steps illustrate how to use the Horizons method to create 3D Mesh stratigraphy.

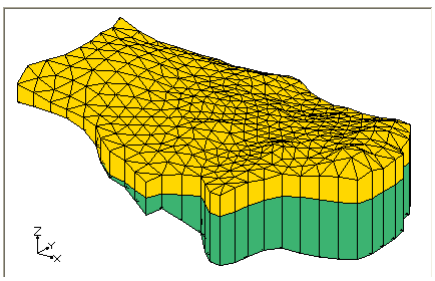
1. **Create/Import Inputs –** There are two main types of inputs for the horizons method:
   1. Boreholes – Boreholes can be created by importing borehole data by using the Text Import Wizard, importing sample data after boreholes already exist, or using the borehole tools to manually enter the boreholes. Once a borehole has been created it can be edited in the Borehole Editor or by using the borehole tools. Also an existing borehole can be copied. Boreholes can be locked to prevent them from being edited.
   2. TINs – TINs can be created 3 different ways in GMS: manually entering the vertex locations and triangulating, converting a different GMS data type to a TIN, and copying a currently existing TIN. (See Creating TINs)
   3. Raster Catalog – a set of rasters defining the top of each horizon (available beginning in version 9.0).
2. **Assign Horizon IDs –** The term “horizon” refers to the top of each stratigraphic unit that will be represented in a corresponding Solid, HUF unit or Material Layer. Horizons are numbered consecutively in the order that the strata are “deposited” (from the bottom up). Horizons can be assigned to both Boreholes and TINs. (See Horizons)
3. **Create Primary 2D Mesh –** A 2D mesh is needed to be used as a projection for the resulting 3D mesh. A 2D mesh or a meshing coverage needs to be created and selected as the primary mesh. The 2D mesh defines the boundary of the 3D Mesh. Also, the meshing options assigned to primary coverage controls the elements of the 3D Mesh that is created. (See Creating 2D Meshes)
4. **Setup additional optional inputs –** Two additional options exist to help constrain and provide user intervention in the Horizon modeling process. The two options are to create borehole cross sections or a horizon conceptual model.
   1. Including Borehole Cross Sections
   2. Horizon Conceptual Model
5. **Run the Horizons Wizard –** Select the ***Horizons→3D Mesh*** command in the *Borehole* or *TINs* menu.

Horizon → 3D Mesh Algorithm

When the Horizon command is executed the horizons specified on the borehole contacts or TIN nodes are converted to a set of scatter points with one dataset for each horizon. The scatter points are then used to interpolate a surface for each horizon. Starting with the lowest numbered horizon, the surface is extruded down to create a set of elements in a 3D Mesh. The surface corresponding to the next horizon is then extruded down to fill in the space between that surface and the previous surface. This process is repeated for each surface. At each step, a set of elements are created for the current horizon and all previously defined elements are subtracted from that layer, resulting in an incremental buildup of the stratigraphy from the bottom to the top. The entire process results in a 3D Mesh with each horizon layer represented by a Material Zone.



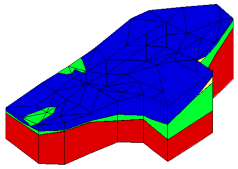
TINs used to create 3D Mesh below



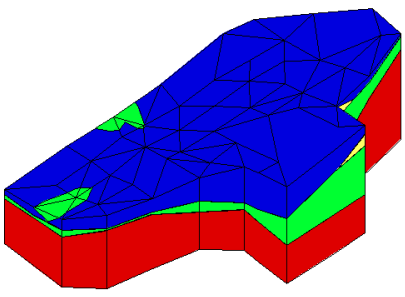
3D Mesh created via the horizons method

5.3. Solid Module

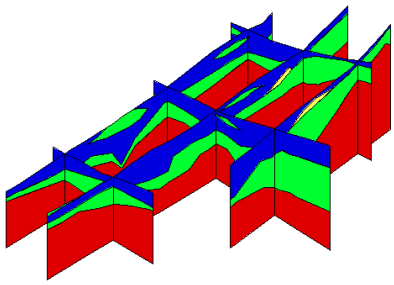
Solid Module

Solid Module

Solid
Solid Properties
Solid Primitives
Solid Cross Sections
More
Boundary Matching
HUF Package
Horizons to Solids
Set Operations
Solid Display Options
Solid Module Tool Palette
Solids to HUF
Solids to Layered Mesh
Solids to MODFLOW Command
Solid Commands

The Solid module of GMS is used to construct three-dimensional models of stratigraphy using solids. Once such a model is created, cross sections can be cut anywhere on the model to create fence diagrams.



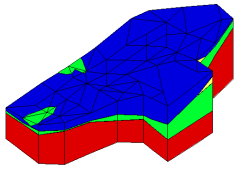
Example of a solid created in GMS



Example of fence diagrams created using solids

Solids are used for site characterization and visualization. Solids can also be used to define layer elevation data for MODFLOW models using the ***Solids → MODFLOW*** command or ***Solids to HUF*** and to define a layered 3D mesh using the ***Solids → Layered Mesh***.

Solid Properties

Solid Module

Solid
Solid Properties
Solid Primitives
Solid Cross Sections
More
Boundary Matching
HUF Package
Horizons to Solids
Set Operations
Solid Display Options
Solid Module Tool Palette
Solids to HUF
Solids to Layered Mesh
Solids to MODFLOW Command
Solid Commands

The Solid properties dialog allows the user to edit/view attributes of the selected solid. This dialog can be accessed by selecting a solid from the Project Explorer, right clicking to access the pop up menu, and selecting the Properties command. It can also be accessed by double clicking on a solid in the graphics window, or by selecting a solid and then selecting the Properties command from the Edit menu.

The following items can be edited in the properties dialog:

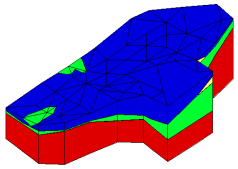
Name	Name of the solid
Material	Material associated with the solid
Begin Layer	beginning grid layer assigned to solid used with Solids→MODFLOW
End Layer	ending grid layer assigned to solid used with Solids→MODFLOW
Use top cell bias	option for using the top cell bias used with Solids→MODFLOW
Top cell bias	the percent to bias the thickness of the top cell with Solids→MODFLOW
Target min. cell thickness	minimum cell thickness used with Solids→MODFLOW

These items are display as information about the solid:

- Solid ID
- Number vertices
- Number triangles

- Max z
- Min z
- Centroid,x
- Centroid,y
- Centroid,z
- Volume

Solid Primitives

Solid Module

Solid
Solid Properties
Solid Primitives
Solid Cross Sections
More
Boundary Matching
HUF Package
Horizons to Solids
Set Operations
Solid Display Options
Solid Module Tool Palette
Solids to HUF
Solids to Layered Mesh
Solids to MODFLOW Command
Solid Commands

To allow the addition of a trench, building, excavation, tunnel, etc. to a solid model, GMS provides the capability of generating several types of simple solid primitives. The solid primitives can be combined using set operations to model man made objects or other subsurface features which cannot be conveniently modeled by extruding TINs.

Cube

Simple cubes or, more precisely, hexahedrons whose faces are all parallel to the x, y, and z planes, can be created by selecting the ***Cube*** command from the *Solids* menu and specifying the center point of the cube and the x, y, and z dimensions.

Sphere

A sphere can be created by selecting the ***Sphere*** command from the *Solids* menu and inputting the radius of the sphere, the coordinates of the centroid of the sphere, and the number of subdivisions. The number of subdivisions determines the density of triangles used to approximate the sphere.

Cylinder

A cylinder can be created by selecting the ***Cylinder*** command from the *Solids* menu and inputting the coordinates of both ends of the cylinder, the radius of the cylinder, and the number of subdivisions in the cylinder. The number of subdivisions determines the density of triangles used to approximate the cylinder. The larger the number, the more accurate the representation will be, however the increased number of triangles will also cause display operations to be slower.

Prism

A prism can be created by first putting the image into plan view and then selecting the ***Prism*** command from the *Solids* menu. The user is then prompted to input a polygon. As with other polygons entered in GMS, the **Backspace** or **Delete key** can be used to delete the last point entered, the **ESC key** can be used to abort the process, and double-clicking terminates point entry. The user is then prompted to enter a bottom elevation and a top elevation for the prism. The default values given for the top and bottom elevation represent elevations just above and just below all of the other solids. The polygon is then extruded from the top to the bottom elevation to create a solid object.

Solid Display Options

The properties of all solid data that GMS displays on the screen can be controlled through the Solids tab of the Display Options dialog. This dialog is opened by right clicking on the ***Solid Data entry*** in the *Project Explorer* and selecting the ***Display Options*** command. It can also be accessed from the from the *Display* menu or the Display Options macro. The following table describes the display options available for the solids module.

Display Option	Description
Solid edges	The Solid edges item is used to display the edges of the solid. The solids are either drawn using the default cell color or the color of the material associated with each solid. The color of the solid edges can be adjusted according to the following options: 1. Auto – draws the material color if faces are not displayed. Uses black or white if the faces are displayed 2. Specified – used the color specified next to the solid edges 3. Material – displays the material color of the solid
Solid faces	The Solid faces item causes the faces of the solid to be drawn as filled polygons.

Solid Module Tool Palette

The following tools are available in the dynamic portion of the Tool Palette whenever the Solid module is activated. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tool in the solid tool palette.

Tool	Tool Name	Description
	Select Solid	The Select Solid tool is used to select solids for deletion or for set operations. When this tool is active, a solid icon appears at the centroid of each solid. A solid is selected by selecting the icon. When a different tool is selected, the icons disappear.
	Select Face	The Select Face tool is used to select the faces of a solid.
	Select Cross Section	Once a set of cross sections has been created, they can be selected using the Select Cross Section tool. Selected cross sections can be deleted or made visible or invisible using the Hide and Show commands. When this tool is active, a cross sections icon appears on each cross section. A cross section is selected by selecting the icon. When a different tool is selected, the icons disappear. When there are several cross sections, it is often easier to differentiate cross section icons in plan view (assuming the cross sections were created in plan view). As a general rule, the icons are placed in the center of the first line segment used to cut the cross section.
	Create Cross Section	Cross sections can be created from the solids that are currently being displayed using the Create Cross Section tool. Cross sections are formed when the user enters a polyline. A polyline is entered by clicking on several points and double-clicking on the final point when the line is finished. The DeleteBackspace key may be used to remove a point from the polyline, and the ESC key can be used to abort the process. A cross section or fence diagram is then computed by cutting perpendicular to the current viewing orientation through the currently visible solids (a solid can purposefully be left out of a cross section by hiding it before making the cross section). A section or "panel" in the fence diagram is created for each line segment in the polyline. While most cross sections are created with the solids in plan view, any viewing orientation can be specified. When cross sections are created, the materials associated with the solids are inherited by cross sections. Cross sections can be saved to a file if desired.

Solids to Layered Mesh

The ***Solids → Layered Mesh*** command in the *Solids* menu can be used to quickly build a 3D finite element mesh that matches the stratigraphy defined by a set of solids. This option is similar to the Boundary Matching option of the ***Solids → MODFLOW*** command except it results in a layered 3D finite element mesh that honors the horizontal boundaries of the stratigraphic layers defined by a set of solids. The steps involved in using the ***Solids → Layered Mesh*** command are as follows:

1. Create a 2D Projection Mesh

The first step is to create a 2D projection mesh. This mesh represents a plan view of the 3D mesh. Each triangle in the 2D mesh will result in a column of 3D wedge elements and each quadrilateral in the 2D mesh will result in a column of 3D hexahedral elements. The 2D mesh can be refined around well locations if desired. This mesh is typically created using the Map module as part of the FEMWATER conceptual model.

2. Create the Solids

The next step is to create the solids defining the stratigraphy. The boundary of the solids should be slightly larger than the 2D projection mesh.

3. Assign layer ranges to the Solids

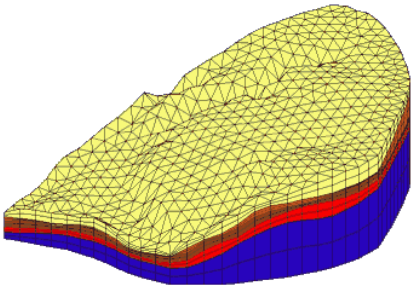
As is the case with the ***Solids → MODFLOW Boundary Matching*** option, a layer range should be assigned to each of the solids. ~~Click here to see some examples of assigning layer ranges.~~ There is one significant difference in assigning layer ranges with the ***Solids → Layered Mesh*** command is that every layer in the 3D mesh must be present in the solids. You can not have "inactive" mesh elements similar to "inactive" grid cells.

4. Solids → Layered Mesh Command

Unlike the ***Solids → MODFLOW Boundary Matching*** option, it is not necessary to create a 3D mesh before selecting the ***Solids → Layered Mesh*** command. The 3D mesh is automatically generated. Each element in the 2D projection mesh is extruded into a vertical column of cells and the solids are used to assign the elevations to the nodes. The material ids are assigned to the 3D elements by finding the solid that encompasses the centroid of each 3D element assigning the material id of that solid to the element.

Sample Mesh

A sample mesh created with the ***Solids → Layered Mesh*** option is shown below. Note that the solids boundaries are preserved at the tops and bottoms of the solids but the transition along the edges of the solids can be irregular.



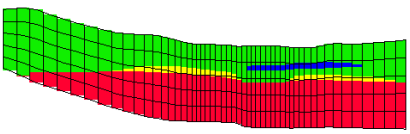
Sample mesh created using the Solids → Layered Mesh command

Solids to HUF

The ***Solids → HUF*** command in the *Solids* menu of the Solid Module can be used to generate the HUF Package input data from the solids currently in your project.

This command brings up the *Solids → HUF* dialog. By turning on the **Adjust grid cell elevations** toggle the user will adjust the 3D grid cell elevations. The **Minimum cell thickness** edit field allows the user to specify a minimum cell thickness. When the elevations are adjusted, if a cell has a thickness less than the minimum the bottom cell in that column will be inactivated and the cell elevations will be recalculated. The **Fraction spreadsheet** contains a row for each layer in the 3D grid. By default, each layer is assigned an equal fraction. The user can edit the fraction assigned to any layer by checking the Edit toggle and changing the fraction. Any layers that don't have the edit toggle checked are evenly distributed so that the sum of the all the fractions is one.

When the user selects OK from the **Solids → HUF** dialog the solids are intersected by the grid. The intersections of the solids are then converted into HUF hydrogeologic unit top and thickness arrays.



Example of HUF data created from solids

Solids to MODFLOW Command

The ***Solids → MODFLOW*** command represents a powerful tool for modeling complex stratigraphy in a completely grid-independent fashion. As part of the overall conceptual model building process, the stratigraphy at a site is modeled as a set of solids. The solids are built using tools in the Borehole, TIN, and Solids modules. These solids can represent a wide variety of complex stratigraphic relationships. The user then assigns hydraulic conductivity (K_h and K_v) and storage coefficients to the solids as material properties and a multi-layer grid is constructed where the boundary of the grid occupies the same region of the solids in plan view. The ***Solids → MODFLOW*** command can then be used to automatically define the elevation arrays in MODFLOW. If the grid is refined or edited in any way, this command can be selected again to rebuild the arrays in seconds with no further user intervention. Together with the feature objects in the Map module, a set of solids can be used to build a completely grid-independent conceptual model regardless of the complexity of the site.

Following is the set of steps required to use the ***Solids → MODFLOW*** command:

1. Building the Solids
2. Material Properties
3. Creating the Grid
4. When using Boundary matching, match layers to solids
5. Execute ***Solids → MODFLOW***

Building the Solids

Before executing the ***Solids → MODFLOW*** command, a set of solids should be contructed that match the site stratigraphy. These solids are typically constructed using the horizons approach. When building these solids, it is best to build the primary TIN with a larger outer polygon boundary than the boundary that is used to define the MODFLOW conceptual model. This ensures that the stratigraphy will encompass all of the grid.

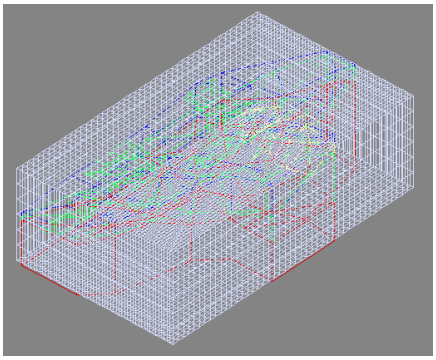
Material Properties

The next step is to create a set of material properties for the solids using the ***Material Properties*** command in the *Solids* menu.

Creating the Grid

Once the solids are created and the layer assignments are made, the next step is to create a grid. The grid boundary in the xy plane (plan view) should either match the boundary of the solids or encompass the solids. The grid can be refined around wells if desired. The number of layers in the grid should be compatible with the layer assignments made to the solids. When the grid is first created, the z elevations can be ignored since they will be inherited from the solids. A sample grid is shown below.

After the grid is created, the cells outside the model domain should be inactivated using the ***Activate Cells in Coverage*** command in *Feature Objects* menu in the Map module.



Sample grid created in GMS

Solids → MODFLOW Options

Now you are ready to execute the ***Solids → MODFLOW*** command. This command brings up a dialog listing the three basic options associated with the ***Solids → MODFLOW*** command. Each option utilizes a different approach for converting the solid stratigraphy to the MODFLOW BCF input arrays. The three options are:

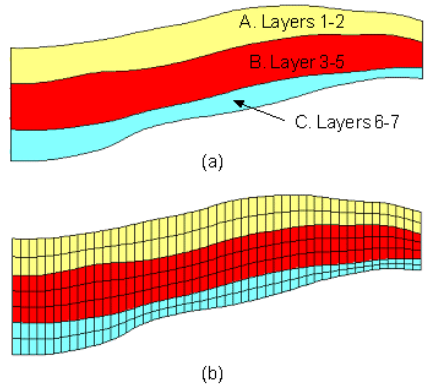
1. Boundary Matching
2. Grid Overlay
3. Grid Overlay with Keq

Boundary Matching

One of the three basic options associated with the ***Solids → MODFLOW*** command is the Boundary Matching option. The goal of the boundary matching algorithm is to compute a set of elevation arrays that honor the boundaries between the stratigraphic units as closely as possible.

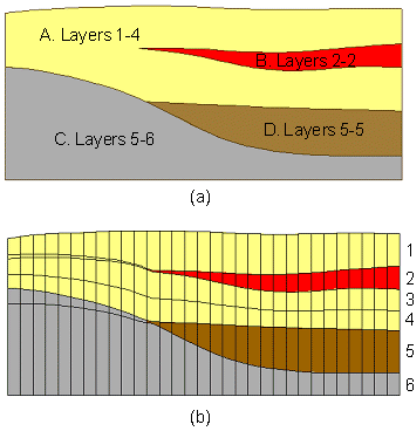
Solids and Layer Ranges

Next a layer range must be assigned to each solid. The layer range represents the consecutive sequence of layer numbers in the MODFLOW grid that are to coincide with the solid model. A sample set of layer range assignments is shown in the figure below (a). The example in the figure below is a case where each solid is continuous through the model domain and there are no pinchouts. Each of the solids is given a layer range defined by a beginning and ending grid layer number. The resulting MODFLOW grid is shown in the figure below (b).



(a) A Set of Simple Solids with Grid Layer Assignments (b) The MODFLOW Grid Resulting From the Layer Assignments

A more complex case with pinchouts is illustrated in the next figure (a). Solid A is given the layer range 1–4, and the enclosed pinchout (solid B) is given the layer range 2–2. The set of grid layers within the defined range that are actually overlapped by the model may change from location to location. The layer range represents the set of grid layers potentially overlapped by the solid anywhere in the model domain. For example, on the left side of the problem shown in the figure below (a), solid A covers grid layers 1, 2, 3 and 4. On the right side of the model, solid A is associated with grid layers 1, 3 and 4 since the enclosed solid (solid B) is associated with layer 2. Likewise, Solid C is associated with grid layers 5 and 6 on the left side of the model but only with layer 6 on the right side of the model where solid D is associated with layer 5. The resulting MODFLOW grid is shown in the figure below (b).



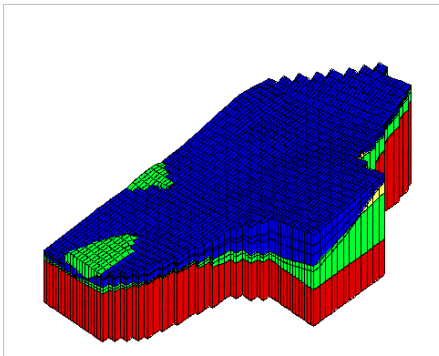
(a) Grid Layer Assignments for a Set of Solids with Pinchouts (b) The MODFLOW Grid Resulting From the Layer Assignments

When assigning layer ranges to solids, care must be taken to define associations that are topologically sound. For example, since solid B in the figure above (a) is enclosed by solid A, solid B could not be assigned a layer range that is outside the layer range of solid A.

Layer ranges are assigned using the *Solids Properties* dialog.

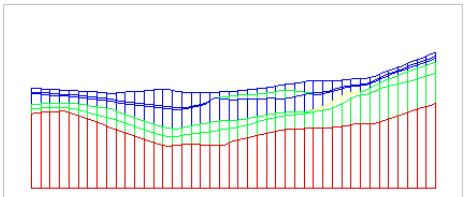
Solids → MODFLOW Command

The final step is to select the ***Solids → MODFLOW*** command in the *Solids* menu. The layer elevations and material properties for the 3D grid will then be automatically assigned from the solids as shown below.

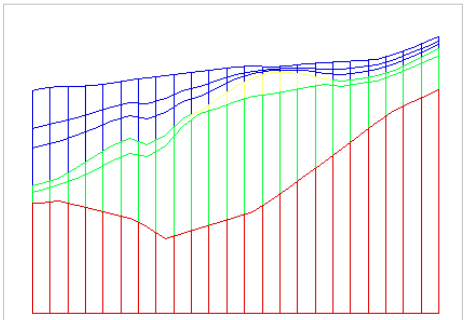


Grid after using the Solids→MODFLOW command

The following images represent cross-sections at selected locations of the grid shown above. Notice that the grid elevations precisely match the stratigraphic boundaries defined by the solids while maintaining the continuous layers required by MODFLOW.



Sample cross-section



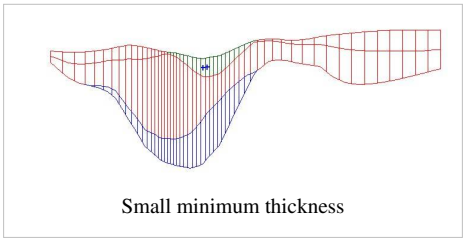
Sample cross-section

Smoothing Tolerance

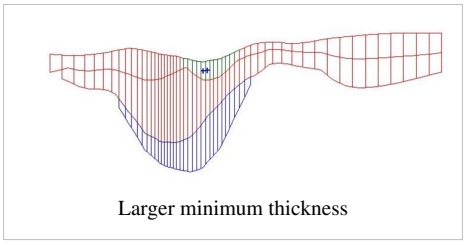
When the ***Solids → MODFLOW*** command is executed with the Boundary Matching option, it is common to have seams that occupy only a portion of a layer as shown in the above cross sections. The top and bottom elevations for cells adjacent to these seams must be adjusted by GMS using a "smoothing" process to ensure that there are not drastic cell size differences in the horizontal direction from one cell to the next. The smoothing is accomplished by iteratively changing the elevation of selected cells until the cell elevations change less than the Smoothing Tolerance specified in the *Solids → MODFLOW Options* dialog.

Minimum Thickness

This option enables users to avoid extremely thin layers at edges of pinchouts and represents the minimum thickness of grid cells created from the solids. Solids with thickness less than this amount are ignored and the surrounding material is used instead. This property is assigned in the solids properties dialog. The figure below demonstrates the application of this property.



Small minimum thickness



Larger minimum thickness

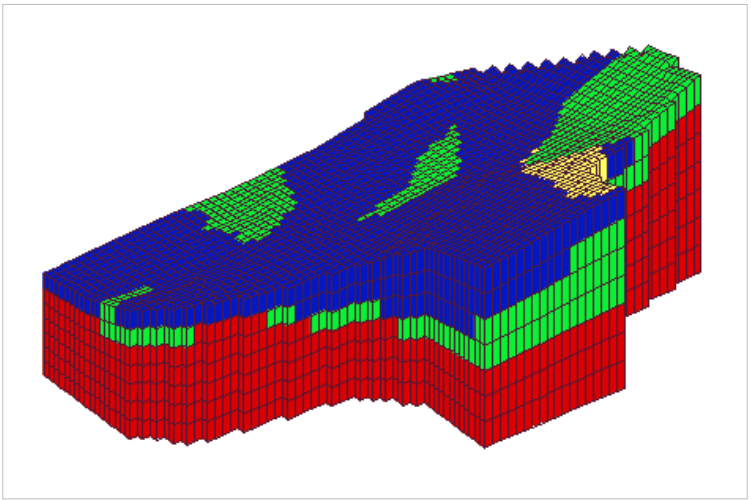
Top Cell Bias

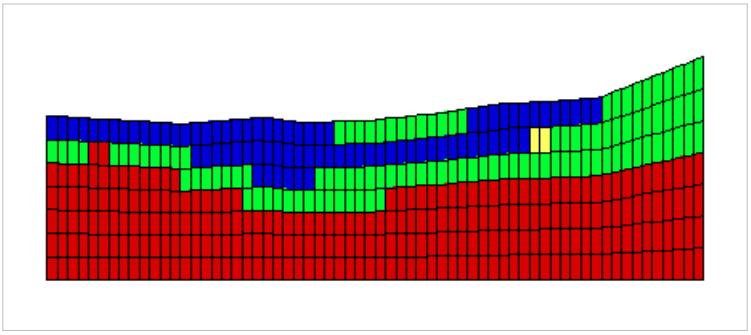
The top cell bias is the percentage of the thickness which is assigned to the top layer of the MODFLOW grid create from the solids. The thickness of the top layer increases as the top cell bias increases. A large top cell bias can be used to prevent top-layer cells from going dry. This property is assigned in the solids properties dialog.

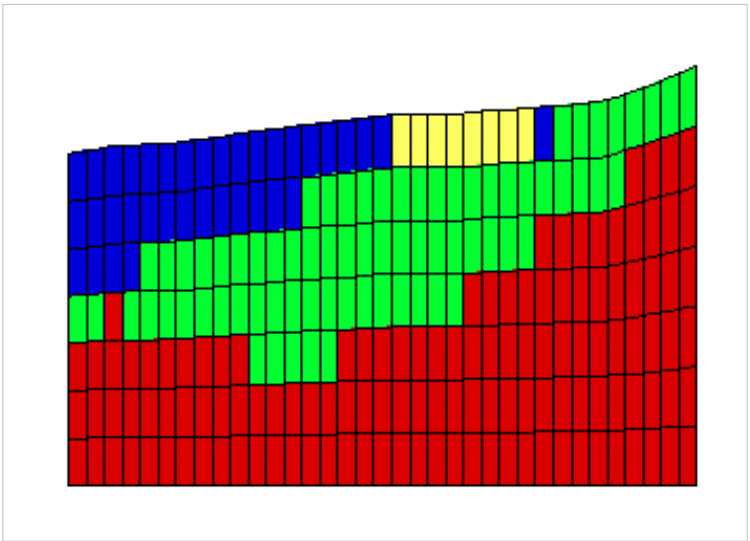
Grid Overlay

The **Grid Overlay** option is one of the three basic options for the ***Solids → MODFLOW*** command. The **Grid Overlay** option is similar to the Boundary Matching option. While the Boundary Matching option precisely matches stratigraphic boundaries, it does have some drawbacks. It can result in very thin layers at certain locations in the grid such as transition points at the boundary of a solid that pinches out to a sharp edge. In some cases, these thin layers can cause stability problems with MODFLOW or with a subsequent transport analysis. For such cases, the Grid Overlay method or the Grid Overlay with Keq method may provide superior results.

With the **Grid Overlay** option, no layer range assignments are necessary. Once the solids and grid are created, the ***Solids → MODFLOW*** command can be immediately selected. For each vertical column of cells, GMS intersects a vertical ray through the cell center and finds the highest and lowest intersection, i.e. the top and bottom of the entire set of solids. These elevations become the top and and bottom elevation of the entire grid. The elevations of any intermediate layer boundaries are then linearly interpolated between these two extremes. The material properties are then assigned by computing the xyz coordinates of the center of each cell and determining which solid encloses the cell center. The material properties from that solid are then assigned to the cell. The result is shown in the following figure (compare this to the example shown in the Boundary Matching topic). Note that the boundaries of the solids are not preserved as accurately as they are with the boundary matching algorithm. However, the cell sizes are much more consistent and extremely thin cells are avoided.







Minimum Thickness

In some cases, the set of solids used with the Grid Overlay method may have thin sections where the vertical thickness of the entire set of solids becomes extremely small. In such cases, the resulting grid cells become very thin as they are "squeezed" in this thin region. The cell thickness in these regions can be controlled using the Minimum Thickness value in the Solids Attributes dialog. When each vertical column of cells is processed, the height of the cells in the column is compared to the minimum thickness. If the cell height is less than the minimum, one or more cells at the bottom of the grid are inactivated until the minimum thickness is satisfied.

Grid Overlay with Keq

The Grid Overlay option is one of the three basic options for the ***Solids -> MODFLOW*** command. This option is very similar to the **Grid Overlay** option. One of the problems with the Grid Overlay option is that if there is a relatively thin layer in the solids and the layer does not happen to encompass any cell centers or it encompasses few cell centers, the layer will be under-represented in the MODFLOW grid. This becomes particularly important if the layer is meant to represent a low permeability layer. For such cases, the **Grid Overlay with Keq** option may give superior results. The **Grid Overlay with Keq** method is identical to the **Grid Overlay** method in terms of how the elevations of the grid cells are defined. The two methods differ in how the material properties are assigned. Rather than simply assigning materials based on which solid encompasses the cell centers, the Keq method attempts to compute a custom K_h and K_v value for each cell. When assigning the material properties to a cell, GMS computes the length of each solid in the cell (from a vertical line at the cell center that intersects the solids) and computes an equivalent K_h, K_v, and storage coefficient for the cell that takes each of the solids in the cell into account. Thus, the effect of a thin seam in a cell would be included in the K_h and K_v values for the cell.

The equivalent K_h is computed as follows:

$$K_h = \frac{\sum K_{hi} M_i}{\sum M_i}$$

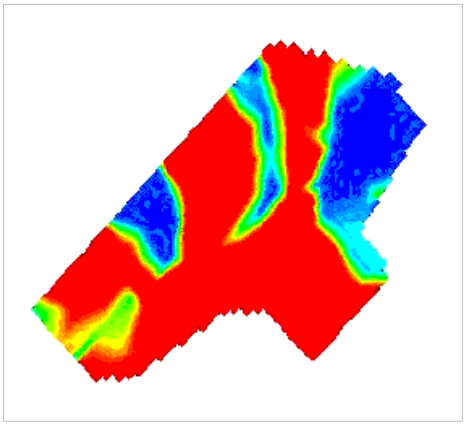
where K_{hi} is the K_h of a solid and M_i is the length of the same solid intersected at the cell center.

The equivalent K_v is computed as follows:

$$K_h = \frac{\sum M_i}{\sum \frac{M_i}{K_{vi}}}$$

where K_{vi} is the K_v of a solid and M_i is the length of the same solid intersected at the cell center.

A horizontal cross-section through a sample grid defined via the Grid Overlay with Keq method is shown below. The colors represent the resulting K_h values. Note how the K values transition at the boundaries of the solids.

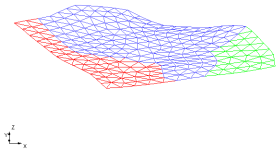


Parameter Estimation

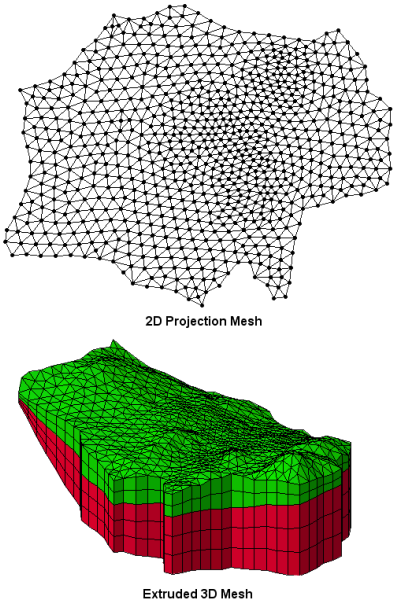
Caution should be taken when using the Grid Overlay with Keq method when performing automated parameter estimation. The "key value" approach to defining parameter zones for PEST requires that the values assigned to the zones be unique within each zone. With the Boundary Matching and Grid Overlay methods, the key values could be assigned to the solids and the values would be properly inherited by the grid cells. With the Grid Overlay with Keq method, the parameter values are "blurred" at the edges of the solids and the key values assigned by the user to the solids would be lost.

5.4. 2D Mesh Module

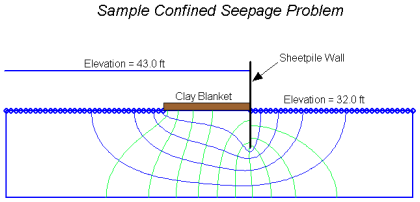
2D Mesh Module

2D Mesh Module

2D Mesh
Creating a 2D Mesh
Editing 2D Meshes
2D Mesh Settings
Converting a 2D Mesh to other types of Data
More
2D Mesh Display Options
2D Mesh Polygon Attributes
2D Mesh Tool Palette
SEEP2D
Solids to Layered Mesh
2D Mesh Commands

The 2D Mesh module is used to construct two-dimensional finite element meshes. Numerous tools are provided for automated mesh generation and mesh editing. 2D meshes are used for SEEP2D modeling and to aid in the construction of 3D meshes. The figures below show an example of a SEEP2D model and a 3D mesh created using the 2D Mesh Module.

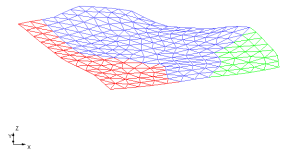


3D Mesh Created by Extruding a 2D Mesh



Sample SEEP2D Problem

Creating a 2D Mesh

2D Mesh Module

2D Mesh
Creating a 2D Mesh
Editing 2D Meshes
2D Mesh Settings
Converting a 2D Mesh to other types of Data
More
2D Mesh Display Options
2D Mesh Polygon Attributes
2D Mesh Tool Palette
SEEP2D
Solids to Layered Mesh
2D Mesh Commands

2D Meshes can be created 3 different ways in GMS: using an automatic meshing technique, manually entering the node locations and triangulating, or converting a different GMS data type to a 2D Mesh.

Using an Automatic Meshing Technique

Map → 2D Mesh is the preferred method for mesh generation in GMS.

Manually Creating a 2D Mesh

In order to create a 2D Mesh in GMS you must have a set of 2D Mesh nodes. Elements can be created by using one of the ***Create Element*** tools in the 2D Mesh Tool Palette and then selecting the mesh nodes to create elements. A 2D Mesh can also be created by triangulating the nodes. The triangulation algorithm assumes that each of the vertices being triangulated is unique in the xy plane, i.e., no two points have the same xy location. Duplicate points can be removed by selecting ***Find Duplicates*** command from the *Mesh* menu. The user is prompted to input a tolerance to be used when checking for duplicate nodes. Two nodes are considered to be duplicates if the XY distance between them is less than or equal to the specified tolerance. The user can also specify whether the duplicate nodes are to be deleted or simply displayed in red.

A 2D Mesh can be created manually from the following steps:

1. Select the ***Create Nodes*** tool from the 2D Mesh Tool Palette.
2. Create the Nodes by clicking inside the Graphics Window at the xy coordinates where you want the vertex located. (To change the node location see Editing 2D Meshes)
3. Select the ***Create Linear Triangle Element*** tool from the 2D Mesh Tool Palette.
4. Select the ***Triangulate*** command from the *Mesh* menu.

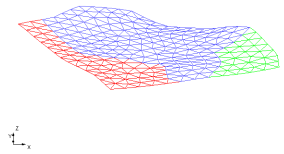
Creating a 2D Mesh from GMS Data

TINs, 2D grids, 2D scatter points, and 3D meshes can all be converted to a 2D Mesh. This is accomplished by using the following commands:

- TIN → 2D Mesh
- 2D Grid → 2D Mesh
- 2D Scatter Points → 2D Mesh Nodes
- ***3D Mesh → 2D Mesh***

After using the ***Scatter Points → Mesh Nodes*** command you must triangulate the nodes to create the 2D Mesh.

Editing 2D Meshes

2D Mesh Module

2D Mesh
Creating a 2D Mesh
Editing 2D Meshes
2D Mesh Settings
Converting a 2D Mesh to other types of Data
More
2D Mesh Display Options
2D Mesh Polygon Attributes
2D Mesh Tool Palette
SEEP2D
Solids to Layered Mesh
2D Mesh Commands

Editing Nodes

Insert Node

New nodes in a 2D mesh are created by selecting the ***Create Nodes*** tool from the 2D Mesh Tool Palette and clicking where the new node is to be located. The default parameters governing the creation of new nodes can be specified using the ***2D Mesh Settings*** command in the *Mesh* menu. This brings up the Node Options dialog.

Delete Node

A set of selected nodes can be deleted by hitting the **Delete key** or selecting the ***Delete*** command from the *Edit* menu. If the deleted node is connected to one or more elements, the action taken when the node is deleted depends on the status of the options in the 2D Mesh Settings dialog.

If the Retriangulate voids when deleting option is turned on, the void created when a node and the elements surrounding the node are deleted is re-triangulated or filled in with triangles. This feature makes it possible to selectively "unrefine" a region of the mesh or reduce the density of the nodes in a region of the mesh without having to completely recreate all of the elements in the region.

If the Retriangulate voids when deleting item in the Node Options dialog is not set, the selected node and the elements surrounding the node are simply deleted and the resulting void is not filled in with triangles.

If the ***Confirm Deletions*** option in the *Edit* menu is active, GMS will prompt the user to confirm each deletion. This feature is helpful in preventing accidental deletions. The Confirm Deletions item is toggled by selecting it from the menu.

Move Node

The coordinates of a 2D Mesh node can be edited by selecting the mesh node and entering the new coordinates in the edit boxes in the Edit Window. It is also possible to drag an existing node to a new location by clicking on the node and moving the mouse with the button held down until the node is in the desired position.

If the ***Snap to Grid*** option in the Drawing Grid Options dialog is set, the node will move in increments corresponding to the drawing grid. If the node being dragged is connected to one or more elements, GMS will not allow the node to be dragged to a position where one of the surrounding elements would become ill-formed.

Since it is possible to accidentally drag points, nodes can be "locked" to prevent them from being dragged by selecting the ***Lock All Nodes*** item from the *Mesh* menu. The nodes can be unlocked by unselecting ***Lock All Nodes*** from the *Mesh* menu.

Editing Elements

Convert Between Linear and Quadratic

Linear elements (three node triangles and four node quadrilaterals) can be converted to quadratic elements (six node triangles and eight node quadrilaterals) and vice versa by selecting the ***Convert Elements*** item from the *Mesh* menu.

If there are both linear and quadratic elements in the mesh (as may be the case with a disjoint mesh), the user is prompted to specify the type of conversion desired, linear to quadratic or quadratic to linear.

Merging Triangles

The triangulate operation creates a mesh composed entirely of triangles. In some cases it is desirable to have the mesh composed primarily of quadrilateral elements. Quadrilateral elements result in a more concise mesh which leads to faster solutions, and quadrilateral elements are often more stable numerically. To address this need, two options are provided for converting triangular elements to quadrilateral elements: The Merge Triangles command, the Merge/Split Tool.

The Merge Triangles Command

The ***Merge Triangles*** command in the *Modify Mesh* menu can be used to automatically merge pairs of adjacent triangular elements into quadrilateral elements. Upon selecting the ***Merge Triangles*** command, the user is prompted to input a minimum interior angle. This angle should be between 0^o and 90^o. If no elements are selected, all of the triangular elements in the mesh are then processed. If some elements have been selected, only the selected elements are processed. The conversion process works as follows:

1. The set of elements to be processed is traversed one element at a time. Each triangular element that is found is compared with each of its three adjacent elements. If the adjacent element is a triangle, the trapezoid formed by the triangle and the adjacent triangle is checked.
2. Each of the four interior angles of the trapezoid is computed and compared to a minimum interior angle. If all of the angles are greater than the user-specified minimum interior angle, then the two triangles are merged into a single quadrilateral element.

This process is repeated for all of the elements. The merging scheme will not always result in a mesh composed entirely of quadrilateral elements. Some triangular elements are often necessary in highly irregular meshes to provide transitions from one region to the next.

The Merge/Split Tool

The other option for merging triangles involves the use of the ***Merge/Split*** tool in the 2D Mesh Tool Palette. This tool can be used to manually merge triangles one pair at a time rather than using the automatic scheme described above.

The manual method is also useful to edit or override the results of the automatic merging scheme in selected areas. The Merge/Split tool can also be used to undo a merge. A quadrilateral element can be split into two triangles by clicking anywhere in the interior of the element. This tool is useful if a pair of triangles is inadvertently merged.

Splitting Quadrilaterals

Occasionally it is necessary to split quadrilateral elements into triangular elements. For example, in order for new nodes to be automatically inserted into a mesh, the elements in the region where the node is inserted must be triangular. Also, in order to process a breakline, the elements in the region of the breakline must be triangular. In such situations, it may be necessary to split a group of quadrilateral elements into triangular elements. Two options are provided for splitting quadrilateral elements:

- **The Split Quads Command –** The ***Split Quads*** command in the *Mesh* menu can be used to split a group of quadrilateral elements into triangular elements. If no elements are selected, all of the quadrilateral elements in the mesh are split. If some elements have been selected, only the selected quadrilateral elements are split.
- **The Merge/Split Tool –** The other option for splitting quadrilateral elements involves the use of the ***Merge/Split*** tool in the 2D Mesh Tool Palette. If the Merge/Split tool is selected, clicking anywhere in the interior of a quadrilateral element with the mouse cursor will cause the element to be split into two triangles. The shortest diagonal through the quadrilateral is chosen as the common edge of the two new triangular elements.

Refining Elements

In some cases, a mesh does not have enough elements in a particular region of the mesh to ensure stability. Rather than inserting supplemental nodes and re-creating the mesh, it is possible to refine a selected region of the mesh using the ***Refine Elements*** command in the *Mesh* menu. This increases the mesh density of a selected area of the mesh. If no elements are selected, the entire mesh is refined. The elevations of the new nodes are interpolated from the existing nodes.

Change Element Materials

Elements can have a material ID associated to it. The ***Materials*** command in the *Edit* menu brings up the *Material Editor* dialog. The material ID associated with the element can be changed using the ***Properties*** command in the *Edit* menu. (See Materials)

Boundary Triangles

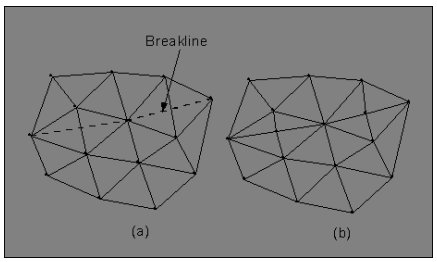
The perimeter of the TIN resulting from the triangulation process corresponds to or approximates the convex hull of the TIN vertices. This may result in some long thin triangles or "slivers" on the perimeter of the triangulated region. There are several ways to deal with the long thin triangles. Thin triangles can be selected and deleted using the normal selection procedures. There is also an option for selecting thin triangles when the Select Triangles tool is selected. If the **Control key** is held down, it is possible to drag a line with the mouse. All triangles intersecting the line are selected. Long thin triangles on the perimeter of the TIN can also be selected by selecting the ***Select Boundary Triangles*** command from the *TINs* menu. The ***Select Boundary Triangles*** command checks triangles on the outer boundary first. If the length ratio of the triangle is less than the critical length ratio, the triangle is selected and the triangles adjacent to the triangle are then checked. The process continues inward until none of the adjacent triangles violate the minimum length ratio. The critical length ratio for selecting thin triangles can be set by selecting the *TINs | TIN Settings* menu command. The length ratio is defined as the longest side of the triangle divided by the

sum of the two shorter sides.

Breaklines

A breakline is a feature line or polyline representing a ridge or some other feature that the user wishes to preserve in a mesh made up of triangular elements. In other words, a breakline is a series of edges to which the triangles should conform to, i.e., not intersect.

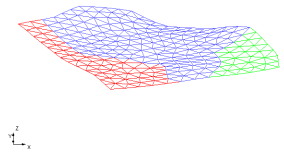
Breaklines can be processed using the ***Add Breaklines*** command from the *Mesh* menu. Before selecting the command, one or more sequences of nodes defining the breakline(s) should be selected using the ***Select Node Strings*** tool in the 2D Mesh Tool Palette.



Breaklines (a) Triangulated Mesh and Breakline. (b) Triangulated Mesh After the Breakline has been Processed.

As each breakline is processed, the triangles intersected by the breakline are modified by adding new nodes at necessary locations to ensure that the edges of the triangles will conform to the breakline. The elevations of the new nodes are based on a linear interpolation of the breakline segments. The locations of the new nodes are determined in such a way that the Delauney criterion is satisfied.

2D Mesh Settings

2D Mesh Module

2D Mesh
Creating a 2D Mesh
Editing 2D Meshes
2D Mesh Settings
Converting a 2D Mesh to other types of Data
More
2D Mesh Display Options
2D Mesh Polygon Attributes
2D Mesh Tool Palette
SEEP2D
Solids to Layered Mesh
2D Mesh Commands

New nodes in a 2D mesh are created by selecting the ***Create Nodes*** tool from the 2D Mesh Tool Palette and clicking where the new node is to be located. The default parameters governing the creation of new nodes can be specified using the ***2DMesh Settings*** command in the *Mesh* menu. This brings up the Node Options dialog. The options in the dialog are as follows:

Default Z

If the check box entitled Interpolate for default z on interior is selected when a new node is inserted in the interior of the mesh, the element enclosing the node is linearly interpolated to get the Z value. If the node is on the exterior of the mesh, the default z value is used. If the toggle is not selected, the default Z is used everywhere.

The options in the center of the dialog are used to specify whether to use a default Z value for all new nodes or to have GMS prompt the user for the Z value every time a new node is created.

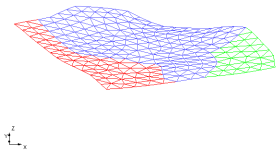
Insert Nodes into Triangulated Mesh

If the check box entitled Insert nodes into triangulated mesh is selected, any new node that lies in a region of the mesh consisting of triangular elements will automatically be incorporated into the mesh. New nodes will not be automatically incorporated into quadrilateral meshes.

Check for Coincident Nodes

If the check box entitled Check for coincident nodes is selected, any new node created using the Create Nodes tool will be checked to see if it lies on top of an existing node.

2D Mesh Display Options

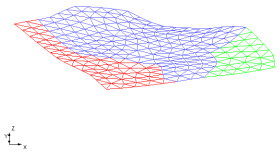
2D Mesh Module

2D Mesh
Creating a 2D Mesh
Editing 2D Meshes
2D Mesh Settings
Converting a 2D Mesh to other types of Data
More
2D Mesh Display Options
2D Mesh Polygon Attributes
2D Mesh Tool Palette
SEEP2D
Solids to Layered Mesh
2D Mesh Commands

The properties of all 2D mesh data that GMS displays on the screen can be controlled through the 2D Mesh tab of the Display Options dialog. This dialog is opened by right clicking on the 2D Mesh Data entry in the Project Explorer and selecting the ***Display Options*** command. It can also be accessed from the from the **Display** menu or the Display Options macro. The following table describes the display options available for the 2D Mesh module.

Display Option	Description
Nodes	The Nodes item is used to display mesh nodes. A small circle is drawn at each node.
Element edges	The Elements item is used to display the edges of elements. The elements can be drawn using either the default color for elements or using the color of the material associated with each element. The color of the element edges can be adjusted according to the following options: 1. Auto – draws the material color if faces are not displayed. Uses black or white if the faces are displayed. 2. Specified – used the color specified next to the cell edges 3. Material – displays the material color of the cell
Element faces	This option fills the elements with the material color.
Texture map image	The Texture Map Image Item is used to "drape" an image over the surface of the 2D Mesh.
Mesh boundary	The Mesh boundary item is used to display a solid line around the perimeter of the mesh. Displaying the boundary is useful when contours are being displayed with the element edges turned off.
Node numbers	The Node Numbers item is used to display the ID associated with each node next to the node.
Element numbers	The Element numbers item is used to display the ID associated with each element at the centroid of the element.

Thin elements	If the Thin elements item is set, triangular elements with small aspect ratios are highlighted. The minimum aspect ratio can be set using the Aspect Ratio command in the Modify Mesh menu.
Scalar values	The Scalar Values item is used to display the scalar values of the active dataset for each node next to the node.
Contours	Most of the objects supported by GMS can be contoured by turning on the Contour Options in the Display Options dialog. When an object is contoured, the values associated with the active data set for the object are used to generate the contours.

2D Mesh Tool Palette

2D Mesh Module

2D Mesh
Creating a 2D Mesh
Editing 2D Meshes
2D Mesh Settings
Converting a 2D Mesh to other types of Data
More
2D Mesh Display Options
2D Mesh Polygon Attributes
2D Mesh Tool Palette
SEEP2D
Solids to Layered Mesh
2D Mesh Commands

The following tools are contained in the dynamic portion of the Tool Palette when the 2D Mesh Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the 2D Mesh tool palette.

Tool	Tool Name	Description
	Select Nodes	The Select Nodes tool is used to select a set of nodes for some subsequent operation such as deletion. The coordinates of a selected node can be edited by dragging the node while this tool is active. The coordinates of selected nodes can also be edited using the Edit Window. A node can also be selected by using the Find Node command in the Mesh menu. The user is prompted for a node ID and the node is selected. Any previously selected nodes are unselected.
	Select Elements	The Select Elements tool is used to select a set of elements for operations such as deletion or assigning a material type. An element can also be selected by using the Find Element command in the Mesh menu. The user is prompted for an element ID and the element is selected. Any previously selected elements are unselected.
	Select Node Strings	The Select Node Strings tool is used to select one or more strings of nodes. Node strings are used for operations such as adding breaklines to the mesh. The procedure for selecting node strings is somewhat different than the normal selection procedure. Strings are selected as follows: • Click on the starting node for the string. The node selected will be highlighted in red. • Click on any subsequent nodes you would like to add to the string (nodes do not have to be adjacent). The selected nodes are now connected by a solid red line. To remove the last node from a string, press the Backspace key. To abort entering a node string, press the ESC key. To end a node string, press Return or double-click on the last node in the string. Another node string can then be selected.
	Create Nodes	The Create Nodes tool is used to manually add nodes to a mesh. When this tool is selected, clicking on a point within the Graphics Window will place a node at that point. What happens to the node after it is added (whether and how it is triangulated into the mesh) depends on the settings in the Node Options dialog in the Modify Mesh menu.

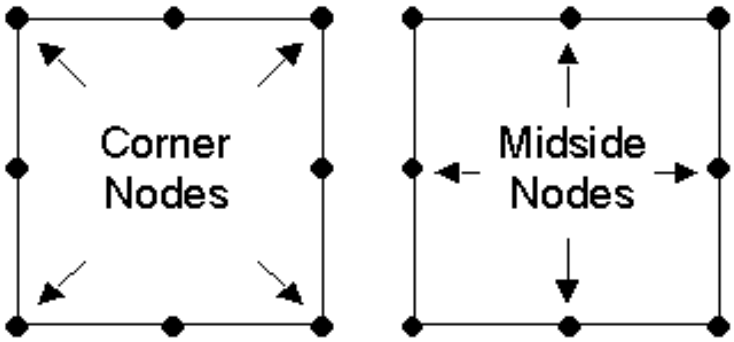
	Create Linear Triangle Element	Four types of elements are supported by the 2D Mesh module: 1. Three node triangles (linear triangles). 2. Six node triangles (quadratic triangles). 3. Four node quadrilaterals (linear quadrilaterals). 4. Eight node quadrilaterals (quadratic quadrilaterals). Elements can be created using automatic meshing techniques such as triangulation. However, it is often necessary to edit a mesh by creating elements one at a time using the four Create Element tools. See the **Creating an Element** section below.
	Create Quadratic Triangle Element	
	Create Linear Quadrilateral Element	
	Create Quadratic Quadrilateral Element	
	Merge/Split	If the Merge/Split tool is selected, clicking on a triangle edge with the mouse cursor will cause the two triangular elements adjacent to the edge to be merged into a quadrilateral element provided that the quadrilateral shape formed by the two triangles is not concave. The Merge/Split tool can also be used to undo a merge or to "unmerge" a quadrilateral element. A quadrilateral element can be split into two triangles by clicking anywhere in the interior of the element. This tool is useful if a pair of triangles are inadvertently merged.
	Swap Edges	If the Swap Edges tool is selected, clicking on the common edge of two adjacent triangles will cause the edge to be swapped as long as the quadrilateral shape formed by the two triangles is not concave. Occasionally, it is useful to interactively or manually swap the edges of two adjacent triangles. This can be thought of as a quick and simple alternative to adding breaklines to ensure that the edges of the triangular elements honor a geometrical feature that needs to be preserved in the mesh.
	Contour Labels	The Contour Label tool is used to manually place numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contour options are changed, until they are deleted using the Contour Labels dialog, or until the mesh is edited in any way. Contour labels can be deleted with this tool by holding down the **Shift** key while clicking on the labels. This tool may only be used when the 2D mesh is in plan view.

Creating an Element

A single element can be constructed from a set of existing nodes using the following steps:

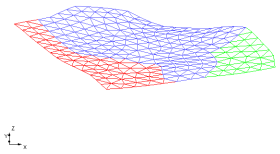
1. Select the tool corresponding to the type of element to be created.
2. Select the nodes corresponding to the corner nodes of the element in consecutive order around the perimeter of the element. The nodes can be selected in either clockwise or counter-clockwise order. It is also possible to build an element by dragging a rectangle to enclose the nodes making up the new element rather than selecting each node one by one. A beep will sound if the wrong number of nodes for the current element type are selected.

If the current element type is a quadratic element (six or eight node element), the midside nodes of the element are created automatically. If the new element is adjacent to an existing element, the midside node of the existing element is used for the new element and a new midside node is not created, i.e. midside nodes are not duplicated.The coordinates of midside nodes cannot be edited. Midside nodes are always assumed to be located at the midpoint of the two adjacent corner nodes. When a corner node is edited, the coordinates of the adjacent midside nodes are updated accordingly.



GMS performs several checks when a new element is constructed. The new element is checked to see whether or not it is ill-formed (the element has a twist in it or is self intersecting). The element is also checked to see if it overlaps any of the elements adjacent to the nodes comprising the new element. In addition, the elements adjacent to a new element are checked to ensure that the elements are conforming, i.e. linear elements (three and four node elements) are not allowed to be placed adjacent to quadratic elements (six and eight node elements). If any of the above checks fail, the construction of the new element is aborted

Converting a 2D Mesh to other types of Data

2D Mesh Module

2D Mesh
Creating a 2D Mesh
Editing 2D Meshes
2D Mesh Settings
Converting a 2D Mesh to other types of Data
More
2D Mesh Display Options
2D Mesh Polygon Attributes
2D Mesh Tool Palette
SEEP2D
Solids to Layered Mesh
2D Mesh Commands

2D Meshes may be converted to other types of data used in GMS, such as a TIN or 2D scatter points. 2D Meshes are converted by using the following commands in the Grid menu:

Mesh → 2D Scatter Points

The ***Mesh → 2D Scatter Points*** command in the *Mesh* menu is used to create a new scatter point set using the nodes in a mesh. A copy is made of each of the datasets associated with the mesh and the datasets are associated with the new scatter point set.

Mesh → TIN

A new TIN can be created from a 2D finite element mesh by selecting the ***Mesh → TIN*** command from the Build *Mesh* menu. A triangle is created from each triangular element in the mesh and two triangles are created from each quadrilateral element in the mesh by splitting the quadrilateral element along the shortest diagonal.

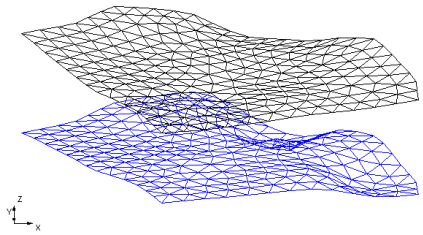
Mesh → 3D Tets

The ***Mesh→3D Tets*** command in the *Mesh* menu is used to convert a 2D mesh to a 3D mesh of tetrahedron.

This command brings up the Mesh→3D Tets dialog. At the top of the dialog the user selects two data sets that will represent the Top elevation and the Bottom elevation of the 3D mesh. In general the top elevation dataset should be completely above the bottom elevation dataset. The user also selects how the 3D mesh will be extruded. There are two options: **Constant number of layers** and **Layers distributed by depth**. When using Constant number of layers, the user simply enters the number of layers in the edit field. When using the Layers distributed by depth, the user enters a maximum layer thickness for each material present in the 2D mesh in the spread sheet.

How it works

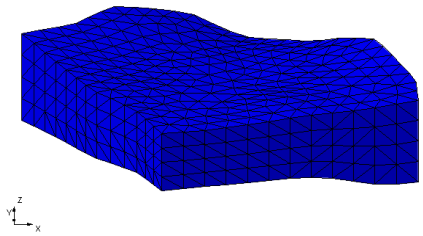
The user specifies a top and bottom elevation dataset and the 3D mesh is extruded between the two datasets. This process is illustrated in the figures below.



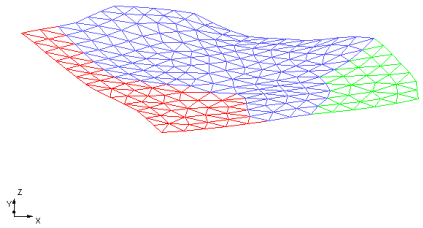
Two surfaces defining the top and bottom of the 3D mesh

Two methods are available for determining the number of 3D mesh nodes to place between the two surfaces: Constant number of layers and Layers distributed by depth.

- The first option is a constant number of layers. The figure above was created by specifying 4 layers between the top and bottom surface. In this case five 3D mesh nodes are created between the top and bottom elevation datasets creating 4 layers.
- The second option uses the materials assigned to the 2D mesh. The user then specifies a maximum layer thickness for each of the materials. Then as the 3D mesh is extruded the number of nodes will vary depending on the material and the distance between the top and bottom elevation. This is illustrated in the figures below. The first figure show a 2D mesh with 3 materials assigned to it.

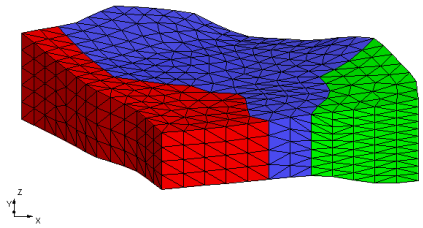


Tet mesh extruded between the top and bottom datasets



2D mesh with 3 materials

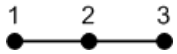
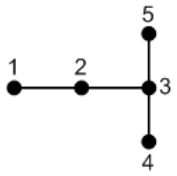
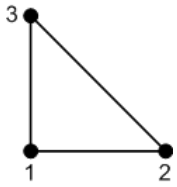
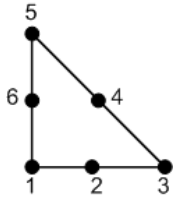
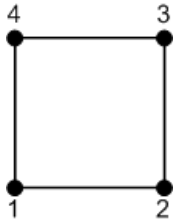
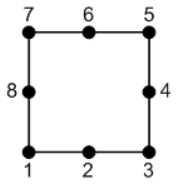
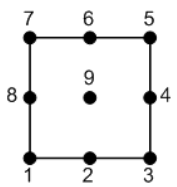
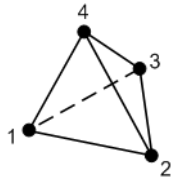
A maximum layer thickness was assigned to each of the materials: red- 14.0 ft., blue- 10.0 ft., and green- 6.0 ft. The resulting 3D mesh is shown below.



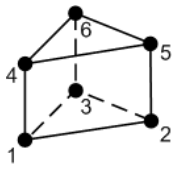
3D tet mesh created with materials and max layer thickness

Element types

Element types used in XMS software. See also XMDF elements [1].

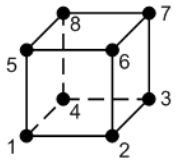
Element Type	Image	Faces
1D linear element with 2 nodes		
1D linear element with 3 nodes		
transition element		
2D linear triangle		
2D quadratic triangle		
2D linear quadrilateral		
2D quadratic quadrilateral		
2D quadratic quadrilateral with center node		
3D linear tetrahedron		<table><tr><th>FaceID</th><th>Node Indices</th></tr><tr><td>1</td><td>2,3,4</td></tr><tr><td>2</td><td>1,4,3</td></tr><tr><td>3</td><td>1,2,4</td></tr><tr><td>4</td><td>1,3,2</td></tr></table>

3D linear prism



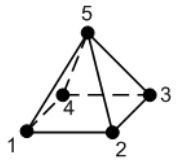
FaceID	Node Indices
1	1,3,2
2	4,5,6
3	1,2,5,4
4	2,3,6,5
5	3,1,4,6

3D linear hexahedron



FaceID	Node Indices
1	1,4,3,1
2	5,6,7,8
3	1,2,6,5
4	2,3,7,6
5	3,4,8,7
6	4,1,5,8

3D linear pyramid



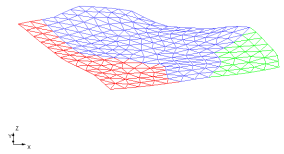
FaceID	Node Indices
1	1,4,3,2
2	1,2,5
3	2,3,5
4	3,4,5
5	4,1,5

Back to XMS

References

[1] http://www.aquaveo.com/xmdf/doc1.6/group__d4d8d2d.html

2D Mesh Polygon Attributes

2D Mesh Module

2D Mesh
Creating a 2D Mesh
Editing 2D Meshes
2D Mesh Settings
Converting a 2D Mesh to other types of Data
More
2D Mesh Display Options
2D Mesh Polygon Attributes
2D Mesh Tool Palette
SEEP2D
Solids to Layered Mesh
2D Mesh Commands

This dialog is used to set the attributes for feature polygons with a SEEP2D or FEMWATER coverage. Attributes that can be specified for each polygon include:

Mesh Type

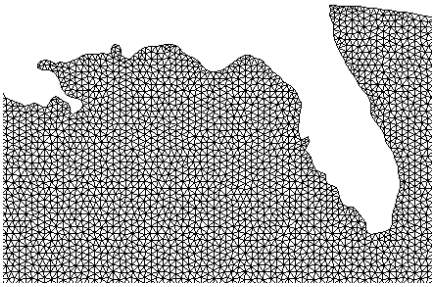
Set the meshing type to be used to fill the interior of the polygon. The options include (different options are given according the coverage type):

None

This results in a hole in the finite element network (no elements are created inside the polygon).

Adaptive Tessellation

Adaptive tessellation is a mesh generation technique used to fill the interior of a polygon. A polygon is assigned to be adaptive tessellation in the polygon attributes dialog and is filled with the Map to 2D Mesh command.


Example mesh created using adaptive tessellation

Adaptive tessellation uses the existing spacing on the polygons to determine the element sizes on the interior. Any interior arcs and refine points are forced into the new mesh. If the input polygon has varying node densities along its perimeter, GMS attempts to create a smooth element size transition between these areas of differing densities. By altering the size bias, the user can indicate whether GMS should favor the creation of large or small elements. Decreasing the bias will result in smaller elements; increasing the bias will result in larger elements. In either case, the elements in the interior of the mesh will honor the arc edges and the element sizes specified at nodes. The bias simply controls the element sizes in the transition region.

Patch

Map Patches

Patching is a mesh generation technique used to fill the interior of a polygon. A polygon is assigned to be a patch in the polygon attributes dialog and is filled with the ***Feature Objects | Map → 2D Mesh*** command.

The coordinates of the new nodes on the interior of the patch are computed by constructing a partial bicubic Coons patch using the polygon as patch edges. This ensures that interior nodes are smoothly interpolated from the nodes making up the perimeter of the patch. Patches are applicable when the data points are gathered along parallel lines, such as cross sections in a river.

Rectangular Patches

The following are some hints when using rectangular patches:

The curvature of the patch can change somewhat, but it should not switch directions. If it does, then the patch should be split at the inflection point of the curve.

Although opposite sides in the rectangular patch are not required to have the same number of nodes, the best patches occur when this is close. In the example shown above, the two ends have the same number of nodes and the two sides only differ by three nodes.

Triangular Patches

All three sides of a triangular patch must have the same number of nodes.

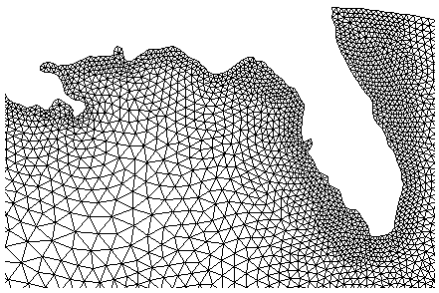
Errors

When the patch is previewed in the polygon attributes dialog, the elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

If a polygon cannot be patched, a help string under the preview window in the polygon attributes dialog explains what needs to be changed.

Paving

With paving the polygon boundary is "paved" inward until the interior is filled. The mesh triangles created from this method are aligned to the boundary.



Example of a mesh created using paving

Polygon Type/Material

Polygons can be assigned a Material type.

Graphical Tools

The Polygon Attributes dialog includes a preview window on the left side. This window shows the arcs and nodes of the selected polygon and allows the user to interact with that definition. The Preview button generates the elements that will be created for the polygon. It is recommended that the preview is used with the patch and adaptive tessellation options only due to the time required performing density meshing. There are several tools for modifying the existing polygon. Zooming, panning and framing work in the preview window just as they normally would in the graphics window of GMS. They are used to facilitate the selection tools.

All entities are selected by clicking on the entity or by dragging a box to select several entities after selecting the tool. The graphical tools are described in the following table.

Tool	Tool Name	Description
	Select Vertex	Select a vertex in the window (red point on arc). Drag the vertex to move it.
	Create Vertex	Create a vertex by clicking on a red arc.
	Select Point/Node	Select a large blue or red node in the window.
	Select Arc	Selects arcs in the preview window.

Arc Options

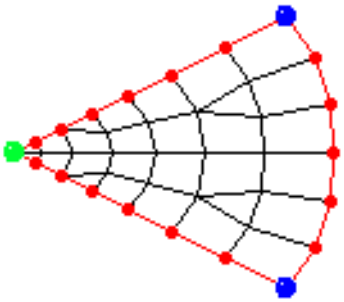
The options dealing with selected arc(s).

- **Use original n nodes**. Use the original vertices on the arc (before entering the dialog). If a vertex is moved, deleted, or created using the above tools, clicking this option will not undo the vertex.
- **Distribute n nodes**. Specify the number of nodes and vertices to put on the arc (minimum of 2). The nodes are evenly spaced if the Bias is 1.0. A Bias of 2.0 will space the nodes more densely to one side; the last space will be twice as big as the first space. A Bias of 0.5 swaps this; the first space is twice as big as the last.

Node Options

The options dealing with selected node(s). These options are used for patches. Patches require 3 or 4 edges. An edge is an arc segment from one blue point to another blue point.

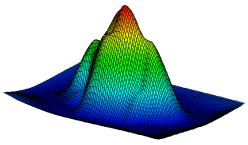
- **Split**. Split two merged arcs. This turns the node blue.
- **Merge**. Merge two arcs. This turns the node red.
- **Degenerate Edge**. This works with 4-sided patches. The degenerate node is treated as an edge, as shown in the figure below.



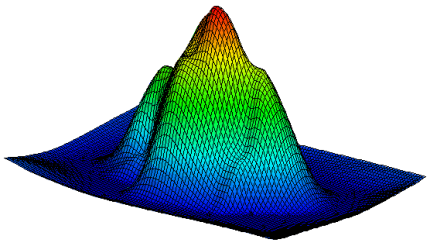
Example of the node options

5.5. 2D Grid Module

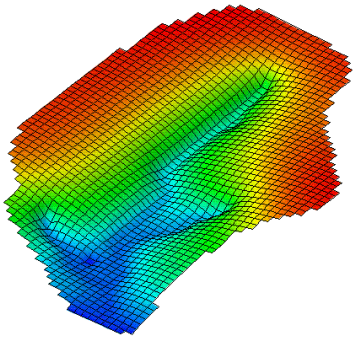
2D Grid Module

2D Grid Module

2D Grid
2D Grid Types
Creating and Editing 2D Grids
Converting 2D Grids
More
2D Grid Display Options
2D Grid Tool Palette
Cell Properties
2D Grid Commands

The 2D Grid module is used for creating and editing two-dimensional Cartesian grids. 2D grids are primarily used for surface visualization and contouring. This is accomplished by interpolating to the grid. The figure below is an example of interpolating contaminant concentration data to a 2D grid.

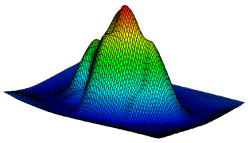


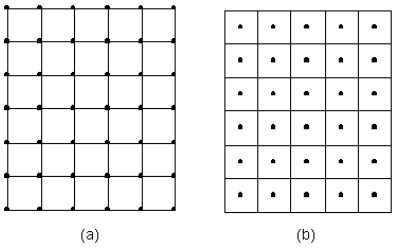
Concentration Data Interpolated to a 2D Grid



Watertable Elevations From a MODFLOW Simulation

2D Grid Types

2D Grid Module

2D Grid
2D Grid Types
Creating and Editing 2D Grids
Converting 2D Grids
More
2D Grid Display Options
2D Grid Tool Palette
Cell Properties
2D Grid Commands



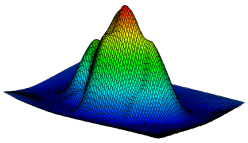
Types of 2D Grids Supported in GMS. (a) Mesh-Centered Grid (b) Cell-Centered Grid.

Two types of grids are supported in the 2D Grid module: mesh-centered grids and cell-centered grids. With a mesh-centered grid, the data values are stored at the corners of the grid cells. With a cell-centered grid, data values are stored at the cell centers.

When a data set is imported to a cell-centered grid, there is one value in the data set for each cell. The contouring and fringing functions use scalar values at the cell corners. Therefore, whenever contouring or fringing is performed, the values at the cell centers are interpolated to the cell corners. Interpolation to cell corners is only done for visualization purposes. All computations performed using the data calculator are performed on the original values at the cell centers. With mesh-centered grids, all visualization and computations are performed at the cell corners and no interpolation is necessary.

Grids in GMS are Cartesian grids. That is, the row and column spacing in the grid can vary, but the row and column boundaries are straight. Each cell center or grid node can have a unique elevation. The grid can also be rotated about the Z axis if desired.

Creating and Editing 2D Grids

2D Grid Module

2D Grid
2D Grid Types
Creating and Editing 2D Grids
Converting 2D Grids
More
2D Grid Display Options
2D Grid Tool Palette
Cell Properties
2D Grid Commands

Creating 2D Grids

Two types of 2D grids are supported by GMS, mesh centered and cell centered. The two main techniques used to create 2D grids are: the ***Create Grid*** command and the ***Map → 2D Grid*** command. A 2D grid can also be created from an existing 3D grid using the ***Grid -> 2D Grid*** command in the *Grid* menu of the 3D Grid Module. A GIS grid may also be imported.

Create Grid

A new grid can be created by selecting the ***Create Grid*** command from the *Grid* menu. This command brings up the Create Grid dialog. The options in the dialog are as follows:

Origin, Length, Rotation – By default, the rows and columns of 2D grids are aligned with the x and y axes. However, grids can be rotated about the z-axis, if desired. Thus, the information needed to determine the overall size and location of the grid is the xy coordinates of the lower left corner of the grid (the lower left corner prior to rotation), the length of the grid in the x and y directions, and the rotation angle. The xy coordinates of the origin are entered in the Origin edit fields, the dimensions are entered in the Length fields, and the angle of rotation is entered in the field entitled Rotation about Z-axis.

Bias – Several options are available for defining the number and locations of the cell boundaries. A bias can be defined which controls how the cell size varies from one cell to the next. For example, an X bias of 1.5 causes each cell to be 50% larger than the previous cell when moving in the positive x direction.

Number of Cells – The total number of cells in each direction (number of rows or columns) can be defined by explicitly entering a number or by entering a base cell size and a limit cell size. The base and limit cell size options are used when a bias other than 1.0 is specified. The base cell size is the size of the first cell in the sequence. The cells are then generated by altering the cell size according to the bias until the limit cell size is reached. The remainder of the cells are constructed using the limit cell size.

Type and Orientation – The controls at the bottom of the Create Grid dialog are used to define the type and orientation of the grid. The user can specify whether the grid should be a mesh-centered grid or a cell-centered

grid. The orientation of the ij axes with respect to the XY axes can also be specified.

Map → 2D Grid

The ***Map → 2D Grid*** command is used to construct a 2D grid using the feature objects in a 2D Grid Coverage. When the ***Map → 2D Grid*** command is selected, the Create Grid dialog appears. If a grid frame has been defined, the size and location of the grid frame are used to initialize the fields in the Create Grid dialog. In most cases, these values will not need to be changed and the user can simply select the OK button to create the grid. If a grid frame has not been defined, the size and location of the grid are initialized so that the grid just surrounds the currently defined feature objects. If desired, the grid dimensions can be edited prior to selecting the OK button to create the grid.

If one or more refine points are defined in the conceptual model, the number of rows and columns in the grid will be automatically determined when the grid is created. Thus, these fields cannot be edited by the user and will be dimmed. If refine points are not defined, the user must enter the number of rows and columns.

Editing 2D Grids

Each of the cells in a 2D grid can be active or inactive. An inactive cell is ignored when contours, fringes, or vectors are displayed on the mesh.

Each cell in the grid has an associated material type. When a new grid is created, the material type for each cell corresponds to the default material type. The default material type can be set using the ***Materials Editor*** command in the *Edit* menu. A new material can be assigned to a cell or a set of cells by selecting the cell(s) and then selecting the ***Properties*** command from the *Edit* menu.

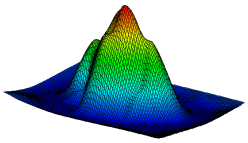
Inserting Rows and Columns

Rows and columns can be added to an existing 2D grid by using the ***Add i Boundary*** tool and the ***Add j Boundary*** tool. Also, the interface between a row and a column can be moved by using the ***Move Boundary*** tool. (See 2D Grid Tool Palette)

Merging Rows and Columns

Rows or columns can be merged together by selecting the rows or columns using the *Select i* or *Select j* tools, right-clicking and selecting the *Merge* command from the pop-up menu. This command is the same as the *Merge Cells* command in the *Grid* menu in the main menu bar.

2D Grid Display Options

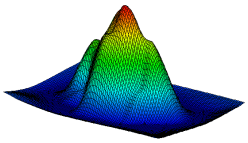
2D Grid Module

2D Grid
2D Grid Types
Creating and Editing 2D Grids
Converting 2D Grids
More
2D Grid Display Options
2D Grid Tool Palette
Cell Properties
2D Grid Commands

The properties of all 2D Grid data that GMS displays on the screen can be controlled through the 2D Grid tab of the Display Options dialog. This dialog is opened by right clicking on the 2D Grid Data entry in the Project Explorer and selecting the ***Display Options*** command. It can also be accessed from the from the *Display* menu or the Display Options macro. The following table describes the display options available for the 2D Grid module.

Display Option	Description
Nodes	The Nodes item is used to display grid nodes depending on the Grid Type . If the grid is cell-centered, a dot is displayed at the cell centers. If the grid is mesh-centered, a dot is displayed on the cell corners.
Cell edges	The Cell edges item is used to display the edges of grid cells. The cells are either drawn using the default cell color or the color of the material associated with each cell. In addition to turning the display of cells on or off, you can temporarily hide grid cells. The color of the cell edges can be adjusted according to the following options: 1. Auto – draws the material color if faces are not displayed. Uses black or white if the faces are displayed. 2. Specified – used the color specified next to the cell edges 3. Material – displays the material color of the cell
Cell faces	The Cell faces item causes the faces of the grid cells to be drawn as filled polygons.
Inactive cells	The Inactive cells item is used to display cells which are inactive. If this option is turned off, inactive cells are not displayed. Inactive cells must be displayed before they can be selected.
Grid boundary	The Grid boundary item is used to display a solid line around the perimeter of the grid. Displaying the boundary is useful when contours are being displayed with the cell edges turned off.
Cell numbers	The Cell Numbers item is used to display the ID of each grid cell.
Node numbers	The Node Numbers item is used to display the ID of each grid node.
Scalar values	The Scalar Values item is used to display the scalar values of the active dataset for each node next to the node.
IJ indices	The IJ indices item is used to display the ij indices of each cell or node.
IJ triad	The IJ triad item is used to display a symbol at one of the corners of the grid showing the orientation of the ij axes.

Texture map image	The Texture map image item is used to "drape" an image over the surface of the 2D Grid.
Contours	Most of the objects supported by GMS can be contoured by turning on the Contour Options in the Display Options dialog. When an object is contoured, the values associated with the active dataset for the object are used to generate the contours.
Vectors	If the Vectors item in the Display Options dialog is selected for an object (TIN, Grid, or Mesh), vector plots can be generated using the active vector dataset for the object. One vector is placed at each node, cell, or vertex.

2D Grid Tool Palette

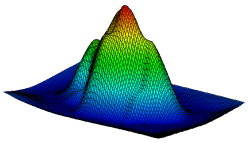
2D Grid Module

2D Grid
2D Grid Types
Creating and Editing 2D Grids
Converting 2D Grids
More
2D Grid Display Options
2D Grid Tool Palette
Cell Properties
2D Grid Commands

The following tools are contained in the dynamic portion of the Tool Palette when the 2D Grid Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the 2D Grid tool palette.

Tool	Tool Name	Description
	Select Cell	The Select Cell tool is used to select individual grid cells or grid nodes. Multi-selection can be performed by holding down the SHIFT key while selecting or by dragging a rectangle to enclose the cells to be selected. The ij indices of the selected cell are displayed in the Edit Window. Only visible cells can be selected. Cells which have been hidden cannot be selected. Inactive cells can only be selected when they are being displayed by turning on the Inactive Cells item in the Display Options dialog. To select specific cell based on the ij of the cell or by cell ID you can use the Find Cell command in the Grid menu. The Find Grid Cell dialog provides edit fields for both an ID or an IJ value. Entering a value for ID will automatically update the IJ fields. Likewise, entering a value for the IJ location will automatically update the ID. When the OK button is selected, the indicated cell will be selected in the grid. In addition to selecting one cell at a time, the Find Grid Cell Dialog can select an entire row column or layer. A zero may be entered in either of the I or J fields indicating that all cells in that direction will be selected. The ID of the cells that will be selected is also displayed as static text at the top of the dialog.
	Select i	The Select i tool is used to select an entire "row" (set of cells with the same i index) of cells at once. Multi-selection can be performed by holding down the SHIFT key. The i index of the selected row is displayed in the Edit Window.
	Select j	The Select j tool is used to select an entire "column" (set of cells with the same j index) of cells at once. Multi-selection can be performed by holding down the SHIFT key. The j index of the selected column is displayed in the Edit Window.
	Select Node	The Select Node tool is used to select nodes and interactively edit cell boundary coordinates by clicking on the intersection of two cell boundaries and dragging the boundaries with the mouse button held down. The coordinates of the cell boundary intersection are displayed in the Edit Window as the boundaries are dragged. If the current view is not the plan view, the dragging movement is constrained to follow the Z axis. The coordinates of a selected boundary intersection can also be edited by directly entering the coordinates in the Edit Window.
	Add i Boundary	The Add i Boundary tool is used to insert a new i boundary into the grid. The new boundary is inserted at the cursor location when the mouse button is clicked. Inserting a new cell boundary changes the dimensions of the grid and all data sets associated with the grid are deleted. If the control key is held down while executing this command, the row will be evenly divided.

	Add j Boundary	The Add j Boundary tool is used to insert a new j boundary into the grid. The new boundary is inserted at the cursor location when the mouse button is clicked. Inserting a new cell boundary changes the dimensions of the grid and all data sets associated with the grid are deleted. If the control key is held down while executing this command, the column will be evenly divided.
	Contour Labels	The Contour Label tool manually places numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contour options are changed, until they are deleted using the Contour Labels dialog, or until the grid is edited in any way. Contour labels can be deleted with this tool by holding down the SHIFT key while clicking on the labels. **This tool can only be used in plan view**.

Converting 2D Grids

2D Grid Module

2D Grid
2D Grid Types
Creating and Editing 2D Grids
Converting 2D Grids
More
2D Grid Display Options
2D Grid Tool Palette
Cell Properties
2D Grid Commands

2D Grids may be converted to other types of data used in GMS, such as a TIN, 2D mesh, or 2D scatter points. 2D Grids can be converted by using the following commands in the *Grid* menu:

Grid → 2D Scatter Points

The ***Grid → 2D Scatter Points*** command in the *Grid* menu is used to create a new scatter point set using the nodes or cells of a 2D grid. A copy is made of each of the datasets associated with the grid and the datasets are associated with the new scatter point set.

Grid → TIN

A new TIN can be created from a 2D grid by selecting the ***Grid → TIN*** command from the *Grid* menu. Two triangles are created from each cell in the grid.

Grid → 2D Mesh

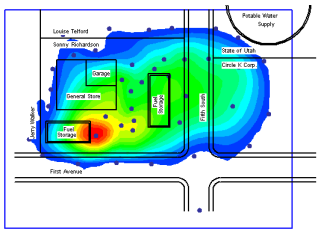
A new 2D finite element mesh can be created from a 2D grid by selecting the ***Grid → 2D Mesh*** command from the *Grid* menu. A four node quadrilateral element is created from each cell in the grid.

Contours → Arcs

Creates a new feature coverage containing arcs derived from the linear contours on the 2D grid. The command only works if linear contours are being displayed.

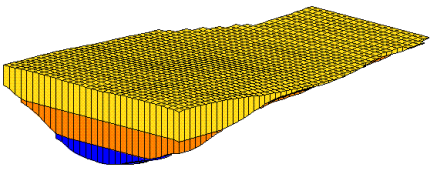
5.6. 2D Scatter Point Module

2D Scatter Point Module

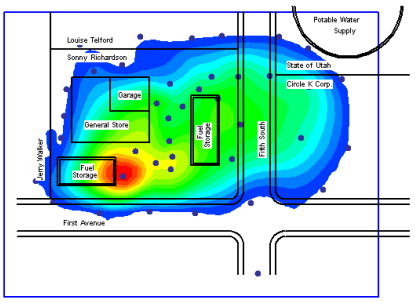
2D Scatter Point Module

2D Scatter Point
Creating and Editing 2D Scatter Point Sets
Interpolating with 2D Scatter Points
Converting 2D Scatter Points to Other Types of Data
More
2D Interpolation Options
2D Scatter Point Display Options
2D Scatter Point Tool Palette
Active/Inactive Points
Clough-Tocher
Inverse Distance Weighted
2D Scatter Point Commands
Interpolation Commands

The 2D Scatter Point module is used to interpolate from groups of 2D scattered data to other objects (meshes, grids, TINs). Several interpolation schemes are supported, including kriging.

Interpolation is useful for setting up input data for analysis codes and for site characterization. The two figures below show examples of using interpolation.

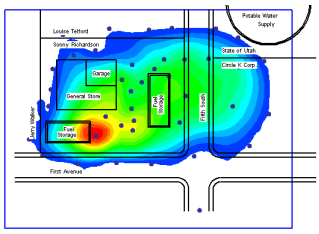


Interpolation of Modflow Layer Elevations to a 3D Grid



Sample Interpolation for Site Characterization

Creating and Editing 2D Scatter Point Sets

2D Scatter Point Module

2D Scatter Point
Creating and Editing 2D Scatter Point Sets
Interpolating with 2D Scatter Points
Converting 2D Scatter Points to Other Types of Data
More
2D Interpolation Options
2D Scatter Point Display Options
2D Scatter Point Tool Palette
Active/Inactive Points
Clough-Tocher
Inverse Distance Weighted
2D Scatter Point Commands
Interpolation Commands

Each of the points from which values are interpolated are called scatter points. A group of scatter points is called a scatter point set. Each of the scatter points is defined by a set of xy coordinates.

Each scatter point set has a list of scalar data sets. Each dataset represents a set of values which can be interpolated to a TIN, mesh, or grid.

Multiple scatter point sets can exist at one time in memory. One of the scatter point sets is always designated as the "active" scatter point set. Interpolation is performed from the active data set of the active scatter point set only.

Creating Scatter Points

Scatter point sets can be created in one of three ways: interactively creating scatter points, converting from other data types, or importing from a file.

Interactively Creating Scatter Points

The Create scatter point tool is used to click out new scatter points in the GMS Graphics Window. The new scatter points are added to the active scatter point set. If you want the new points in their own scatter point set then select the ***Scatter Points | New Scatter Point Set*** command.

Converting from Other Types

Scatter point sets are often created by converting from other data types (TINs, meshes, grids, boreholes). The following commands are available to convert an object to a Scatter Point Set:

- TIN → 2D Scatter_Points
- Contacts → 2D Scatter Points
- Watertable → 2D Scatter Points
- Grid → 2D Scatter Points
- Mesh → 2D Scatter_Points
- Map → 2D Scatter Points
- Modflow Layers → 2D Scatter_Points

Importing Tabular Scatter Point Data

In most cases, scatter point sets are created by importing a text file through the Text Import Wizard.

Editing Scatter Points

The location of a scatter point can be edited by selecting the scatter point and dragging it to a new location or by typing in the new coordinates in the Edit Window.

The data set value associated with a scatter point can be edited using the edit field labeled "F:" in the Edit Window. Data set values can also be edited using a spreadsheet dialog by selecting the Edit Values button in the Dataset Info dialog.

In addition to the dataset values, each scatter point has three properties that can be edited on a point by point basis:

- label
- material
- activity
- Fixed pilot point

The label is a text string that can be displayed by turning on the ID option in the Display Options dialog. The material type is used for indicator simulations.

The fixed pilot point property is used with PEST(See Automated Parameter Estimation). If this option is on, then the value at this point is not estimated during the parameter estimation process.

The scatter point properties can be edited by **double-clicking** on a point or by selecting a set of points and selecting the ***Properties*** command in the *Edit* menu.

Editing Scatter Points

Individual scatter points can also be deleted. This command results in the removal of the point from all the datasets associated to the scatter point set.

2D Scatter Point Display Options

The properties of all 2D scatter data that GMS displays on the screen can be controlled through the 2D Scatter tab of the Display Options dialog. This dialog is opened by right clicking on the 2D Scatter Data entry in the Project Explorer and selecting the ***Display Options*** command. It can also be accessed from the from the *Display* menu or the Display Options macro. The following table describes the display options available for the 2D Scatter Point module.

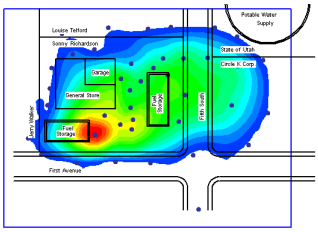
Display Option	Description
Active scatter point set	The name of the active scatter point set is listed at the top of the dialog. The symbol selected using the Scatter point symbols option (described below) applies to the active scatter point set. This makes it possible to use a different set of symbols for the points in each set so that the sets are easily distinguishable.
Scatter point symbols	The Scatter point symbols item is used to display a symbol at the location of each scatter point. The widget to the left of the toggle is used to bring up a dialog listing the available symbols. The color of each of the scatter points in a set may be changed in this dialog also. The color of the scatter points can be adjusted according to the following options: 1. Specified – uses the color specified next to the scatter point symbols 2. Data – the color ramp is used to assign a color to each of the symbols according to the value of the active scalar dataset
Contours	Individual scatter points can be colored based on the current dataset by using contour options.
Inactive scatter points	Individual scatter points can either be active or inactive. The Inactive scatter points option can be used to control the display of the inactive points.
Scatter point scalar values	The Scatter point scalar values option is used to display the value of the active data set next to each of the scatter points.
Scatter point labels	The Scatter point labels item is used to display the scatter point label next to each scatter point.
Scatter point numbers	The Scatter point IDs item is used to display the scatter point ID next to each scatter point.
Symbol legend	The Symbol legend item is used to display a symbol legend listing each of the scatter point sets by name and showing the symbol associated with the scatter point sets.

2D Scatter Point Tool Palette

The following tools are active in the dynamic portion of the Tool Palette whenever the 2D Scatter Point Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window with the cursor depends on the current tool. The table below describes the tools in the 2D Scatter Set tool palette.

Tool	Tool Name	Description
	Select Scatter Point	The Select Scatter Point tool is used to select individual scatter points for editing using the Edit Window. Scatter points can also be dragged with the mouse. Scatter points can be deleted. With extremely large sets of scatter points, it may become difficult to identify a scatter point with a particular ID, even if the scatter point IDs are being displayed. In such cases, the Find Point command in the Scatter Points menu can be used to quickly locate a point. The command prompts the user for the ID of the desired point and the point is selected.
	Select Scatter Point Set	The Select Scatter Point Set tool is used to select entire scatter point sets for deletion or to designate the active scatter point set. When this tool is active, an icon appears at the centroid of the set for each of the scatter point sets. A scatter point set is selected by selecting the icon for the set.
	Create Scatter Point	This tool is used to interactively create scatter points by clicking in the GMS graphics window.

Interpolating with 2D Scatter Points

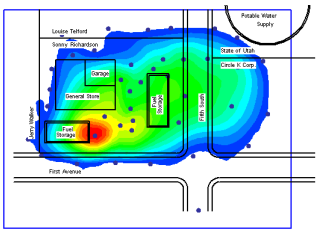
2D Scatter Point Module

2D Scatter Point
Creating and Editing 2D Scatter Point Sets
Interpolating with 2D Scatter Points
Converting 2D Scatter Points to Other Types of Data
More
2D Interpolation Options
2D Scatter Point Display Options
2D Scatter Point Tool Palette
Active/Inactive Points
Clough-Tocher
Inverse Distance Weighted
2D Scatter Point Commands
Interpolation Commands

Scatter point sets are used for interpolation to other data types such as TINs, grids, and meshes. A 2D grid can be created which will just enclose the scatter points by using the ***Bounding Grid*** command in the *Scatter Points* menu. Interpolation is useful for such tasks as contouring or setting up input data to a model. Since no interpolation scheme is superior in all cases, several interpolation techniques are provided in GMS.

The basic approach to performing an interpolation is to select an appropriate interpolation scheme and interpolation parameters, and then interpolate to the desired object using one of the 2D interpolation commands.

The interpolation options are selected using the Interpolation Options dialog accessed through the ***Interp. Options*** command in the *Interpolation* menu. Once a set of options is selected, those options are used for all subsequent interpolation commands.

Converting 2D Scatter Points to Other Types of Data

2D Scatter Point Module

2D Scatter Point
Creating and Editing 2D Scatter Point Sets
Interpolating with 2D Scatter Points
Converting 2D Scatter Points to Other Types of Data
More
2D Interpolation Options
2D Scatter Point Display Options
2D Scatter Point Tool Palette
Active/Inactive Points
Clough-Tocher
Inverse Distance Weighted
2D Scatter Point Commands
Interpolation Commands

2D Scatter Points may be converted to other types of data used in GMS, such as a TIN, 2D Mesh Nodes, or Observation Points. 2D Scatter Points are converted by using the following commands in the Scatter Points menu:

- *Scatter Points → TIN*

  The Scatter Points → TIN command creates a set of TIN vertices. These vertices are automatically triangulated to form a TIN.

- *Scatter Points → Mesh Nodes*

  The Scatter Points → Mesh Nodes command creates a set of 2D finite element nodes from the points in the active scatter point set.

- *Scatter Points → Obs. Pts.*

  The Scatter Points → Obs. Pts command creates one observation point for each of the scatter points in the active scatter point set. The active dataset values become the measured values for the observation points. You must create a coverage with a measurement before executing this command.

- *Scatter Points → 3D Scatter Points*

  The Scatter Points → 3D Scatter Points command creates a 3D scatter point set from the 2D scatter points. All of the datasets are copied to the 3D scatter points. The Z (elevation) of the 3D scatter points is set to the Z of the 2D scatter points at the time of conversion.

Gaussian Field Generator

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

GMS includes an interpolation option associated with the 2D scatter point module called Gaussian Sequential Simulation (GSS). This option is used to generate a set of scalar datasets (Gaussian fields) using a Gaussian sequential simulation. This is somewhat similar to indicator kriging or T-PROGS in that it generates a set of equally probable results which exhibit heterogeneity and are conditioned to values at scatter points. However, the resulting arrays are floating point scalar datasets, rather than the integer arrays produced by T-PROGS and indicator kriging.

The results of a GSS can be used in combination with the new Multiplier Array option for parameters. It is now possible to associate one or more scalar datasets with an array-based parameter. When MODFLOW is executed, the parameter starting value is multiplied by the dataset to produce the input array. This makes it possible to use the results of the Gaussian sequential simulation as input for parameter fields for a stochastic (Monte Carlo) simulation.

Gaussian Simulations

The new GSS tool is based on the FIELDGEN code developed by John Doherty. John Doherty describes GSS as follows:

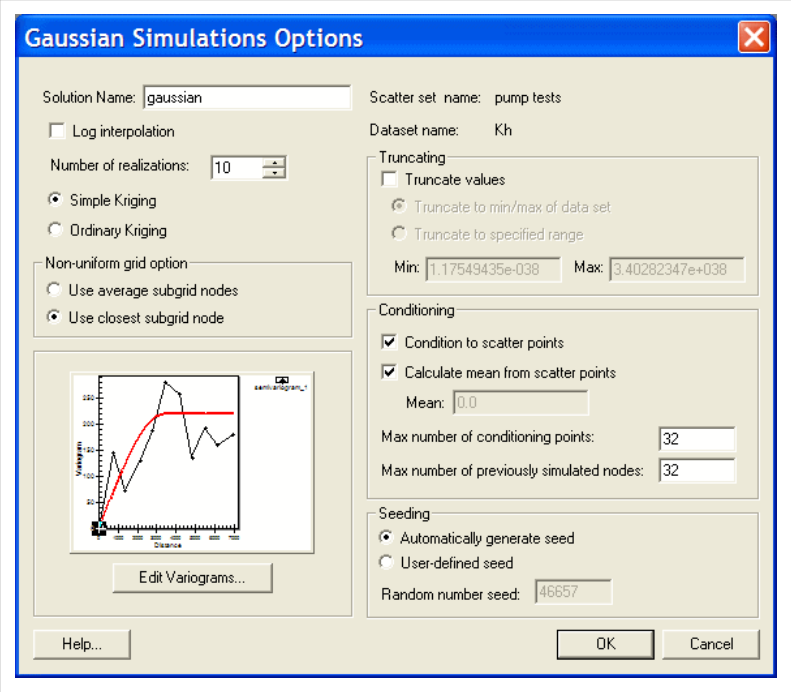
The process of stochastic field generation by sequential simulation is very easy to understand. At each field point an expected field value and a field standard deviation pertaining to that point are first determined. These are calculated through kriging from points to which field values have already been assigned, as well as from points at which conditioning data exists (if available). Using the expected value and standard deviation calculated in this way, a random field value is generated based on the assumption of a Gaussian probability distribution. The field value thus obtained can then be used in generating expected values and standard deviations at other field points at which field generation then takes place in the same way.

GSS is a form of Kriging but it is listed in the GMS interface as a new interpolation scheme. This new option will differ from Kriging in the following ways:

1. GSS uses the FIELDGEN utility developed by John Doherty to perform the interpolation rather than the GSLIB code used by kriging. FIELDGEN is a modified version of the **sgsim** utility in GSLIB so many of the options are quite similar to those used for normal kriging.
2. As is the case with T-PROGS, the user enters the number of desired simulations and FIELDGEN produces N arrays, rather than one array.
3. It can only be used for 2D interpolation and it will only work when interpolating to 3D cell-centered grids.
4. It can work with or without a scatter point set. If a scatter point set is provided, the resulting fields are conditioned to the values at the scatter points. Otherwise the user defines a mean and a variogram and the values are randomly generated.

Gaussian Simulation Options

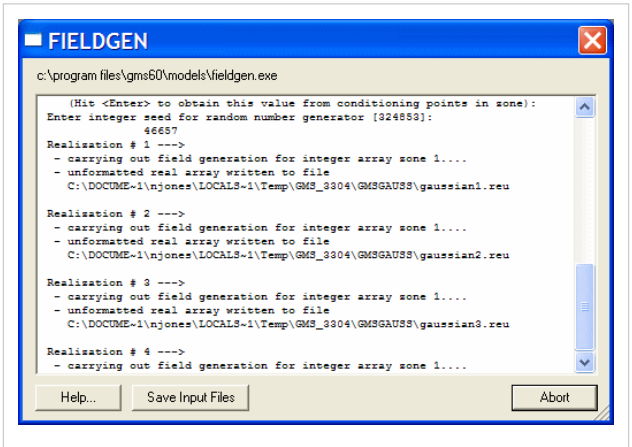
The first step in setting up a GSS is to import a set of scatter points with the values to which you intend to condition your simulation. This step can be skipped if you have no conditioning data. The next step is to select the ***Gaussian Simulation Options*** command in the *Interpolation* menu in the 2D Scatter Point module. This brings up the following dialog:



The **Solution name** at the top is the name that will be applied to the set of Gaussian fields. The **Number of realizations** item is the desired number of Gaussian fields. The original GSLIB code was designed to work with uniform grids (constant cell sizes). The **Non-uniform grid** option controls how the data are converted to a non-uniform grid (if necessary). The **Edit Variogram** button should be selected to set up a model variogram using the GMS variogram editor. A model variogram must be defined whether or not you have scatter points for conditioning.

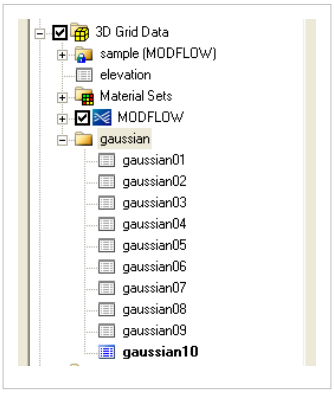
Running the Simulation

Once the GSS options are selected, the next step is to run the simulation. This is accomplished by selecting the ***Run Gaussian Simulation*** command in the *Interpolation* menu. During the simulation, you should see a window displaying the progress of the simulation:

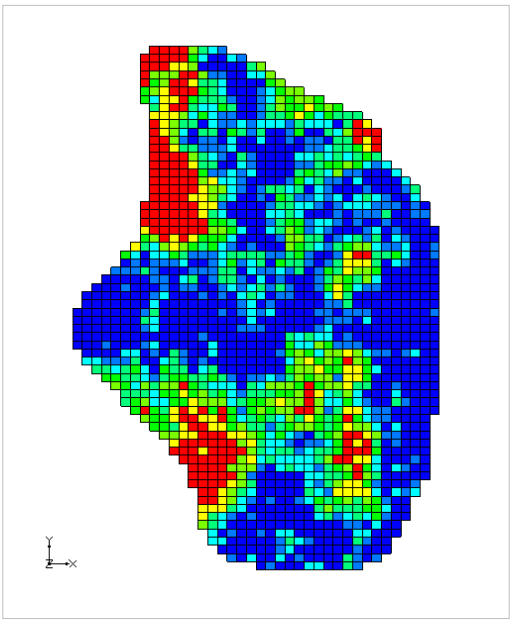


Viewing the Results

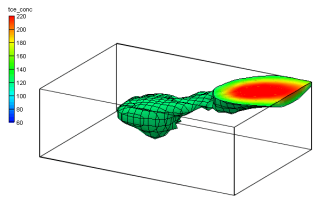
Once the simulation is finished, you should see a new folder appear in the Project Explorer window which has the name of the simulation and contains a set of dataset arrays:



Clicking on each data set icon makes it the active dataset for contouring. The dataset properties can be viewed by double-clicking on the icon. The following image represents a sample Gaussian realization:


Y
Z
X

Active/Inactive Points

3D Scatter Point Module

3D Scatter Point
Interpolating with 3D Scatter Points
Converting 3D Scatter Points to Other Data Types
Bounding Grid
More
3D Scatter Point Display Options
3D Scatter Point Tool Palette
Active/Inactive Points
Inverse Distance Weighted
3D Scatter Point Commands

Each scatter point has an active/inactive status. A scatter point with an inactive status can be displayed, but the dataset value at the point is ignored when interpolation takes place. As a result, interpolation proceeds as if the point did not exist.

The active/inactive flags for scatter points are particularly useful when dealing with transient data. For example, suppose that a set of scatter points represents TCE concentrations measured at a series of observation wells over a year's time. The locations of the wells and the measured concentrations can be imported to GMS as a scatter point set with a transient dataset. Once they are imported, the transient dataset can be interpolated to a grid and a film loop showing color shaded contours can be generated to illustrate how the plume has changed with time. However, in preparing the data for import, it is discovered that some of the data values are missing. One approach is to make up a dummy value for the missing sample and enter the entire dataset anyway. The problem with this approach is that it is difficult to determine an appropriate dummy value. Another option is to enter this value as a "non-detect". This causes the point to become inactive for the time step where the sample is missing. GMS disregards the point for that time step and performs the interpolation using the remaining active points.

Active/inactive flags are stored with datasets. If the active data set is changed, the active/inactive flags will be reassigned based on the flags in the new active dataset. Not all data sets contain active/inactive flags. If a data set does not contain flags, all points are assumed to be active.

The following methods can be used for controlling or assigning the active/inactive status of points:

Tabular Scatter Point Input

If the scatter points are imported using the Text Import Wizard then a special data value can be designated as NONDETECT. This value is typically assigned to a number not likely to be encountered such as -999. Then, as the data set columns are being read, any value with the NONDETECT value is assumed to be inactive and the status flag is set accordingly.

Active/Inactive Flags Dialog

After a scatter point set has been imported to GMS, the active/inactive status flags for the active dataset can be edited by selecting the Edit Inactive Flags button in the Dataset Info dialog accessed from the Dataset Properties dialog (this dialog is accessed from the Project Explorer). This brings up the Active/Inactive Flags dialog. This dialog is used to either delete all of the current active/inactive flags (making all points active), or enter one or more key values (ex., -999) which are used to inactivate all points with the listed values.

2D Interpolation Options

The interpolation options are selected using the Interpolation Options dialog accessed through the ***Interp. Options*** command in the *Interpolation* menu. Once a set of options is selected, those options are used for all subsequent interpolation commands. The items in the 2D Interpolation Options dialog are as follows:

Active Dataset

Interpolation is always performed using the active dataset of the active scatter point set. The active dataset is normally selected in the Project Explorer. The name of the current active dataset is listed at the top of the 2D Interpolation Options dialog. The active dataset can not be changed with this dialog.

If the active dataset is transient then more interpolation options are available. (see Steady State vs. Transient Interpolation)

Interpolation Method

The following 2D interpolation methods are supported by GMS:

- Linear
- Inverse Distance Weighted
- Clough-Tocher
- Natural Neighbor
- Kriging

Log Interpolation

When interpolating chemical data, it is not uncommon to have a small "hot spot" somewhere in the interior of the data where the measured concentrations are many orders of magnitude higher than the majority of the other concentrations. In such cases, the large values dominate the interpolation process and details and variations in the low concentration zones are obliterated. One approach to dealing with such situations is to use log interpolation. If this option is selected, GMS takes the log of each data value in the active scatter point set prior to performing interpolation. By interpolating the log of the dataset, small values are given more weight than otherwise. Once the interpolation is finished, GMS takes the anti-log (10^x) of the interpolated dataset values before assigning the dataset to the target grid or mesh.

Note that it is impossible to take the log of a zero or negative value. When the log interpolation option is turned on, a value must be entered by the user to assign to scatter points where the current data value is less than or equal to zero. Typically, a small positive number should be used.

Anisotropy

Sometimes the data associated with a scatter point set will have directional tendencies. The azimuth and horizontal anisotropy allow the user to take into account these tendencies.

Extrapolation

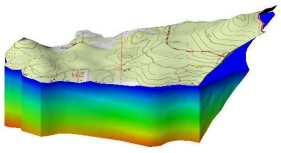
Although they are referred to as interpolation schemes, most of the schemes supported by GMS perform both interpolation and extrapolation. That is, they can estimate a value at points both inside and outside the convex hull of the scatter point set. Obviously, the interpolated values are more accurate than the extrapolated values. Nevertheless, it is often necessary to perform extrapolation. Some of the schemes, however, perform interpolation but cannot be used for extrapolation. These schemes include Linear and Clough-Tocher interpolation. Both of these schemes only interpolate within the convex hull of the scatter points. Interpolation points outside the convex hull are assigned the ***Default extrapolation value***.

Truncation

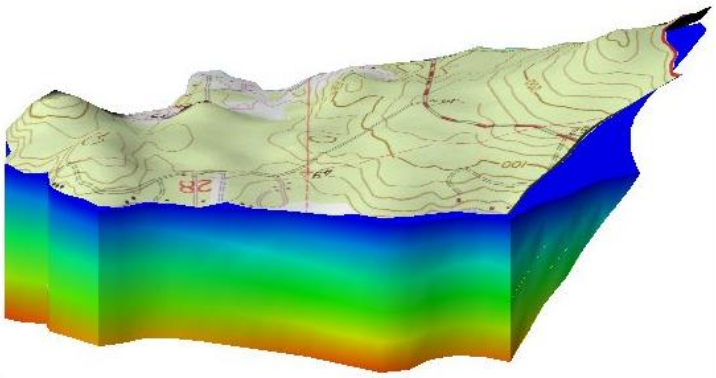
When interpolating a set of values, it is sometimes useful to limit the interpolated values to lie between a minimum and maximum value. For example, when interpolating contaminant concentrations, a negative value of concentration is meaningless. However, many interpolation schemes will produce negative values even if all of the scatter points have positive data values. This occurs in areas where the trend in the data is toward a zero value. The interpolation may extend the trend beyond a zero value into the negative range. In such cases it is useful to limit the minimum interpolated value to zero. Interpolated values can be limited to a given range by selecting the Truncate values option in the Interpolation Options dialog. The range can be user-defined or automatically set to the maximum and minimum values of the dataset being interpolated.

5.7. 3D Mesh Module

3D Mesh Module

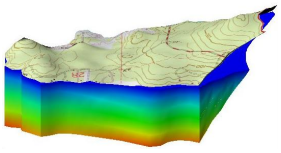
3D Mesh Module

3D Mesh
3D Element Types
Creating a 3D Mesh
Editing a 3D Mesh
Converting 3D Meshes to Other Data Types
More
3D Mesh Display Options
3D Mesh Tool Palette
ADH Display Options
Solids to Layered Mesh
3D Mesh Commands

The 3D Mesh module is used to create and edit 3D finite element meshes. Once a mesh is constructed, the FEMWATER interface can be used to assign boundary conditions and analysis parameters and perform a FEMWATER analysis.

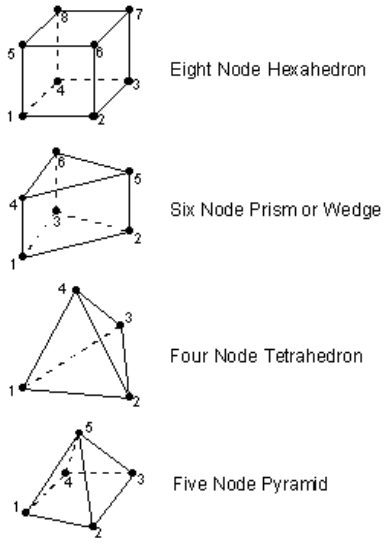


Iso-surface of Pressure Head from a FEMWATER Simulation

3D Element Types

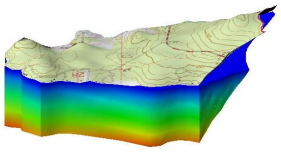
3D Mesh Module

3D Mesh
3D Element Types
Creating a 3D Mesh
Editing a 3D Mesh
Converting 3D Meshes to Other Data Types
More
3D Mesh Display Options
3D Mesh Tool Palette
ADH Display Options
Solids to Layered Mesh
3D Mesh Commands

Four types of 3D elements are supported by GMS: eight node hexahedra, six node prisms or wedges, four node tetrahedra, and five node pyramids. Hexahedra and wedges are created by projecting a 2D mesh. Tetrahedral elements are constructed with the ***Tessellate*** command or they can be created elsewhere and imported into GMS.



Types of 3D Elements Supported by GMS

Creating a 3D Mesh

3D Mesh Module

3D Mesh
3D Element Types
Creating a 3D Mesh
Editing a 3D Mesh
Converting 3D Meshes to Other Data Types
More
3D Mesh Display Options
3D Mesh Tool Palette
ADH Display Options
Solids to Layered Mesh
3D Mesh Commands

In order to create a 3D Mesh in GMS you must have a set of 3D Mesh nodes. Elements can be created by using one of the create element tools and then selecting the mesh nodes to create elements. Duplicate points can be removed by selecting ***Find Duplicates*** command from the *Mesh* menu. If a node is found that is within a user specified tolerance of another node, the node is either selected or deleted.

3D Meshes can be created 2 different ways in GMS: converting a different GMS data type to a 3D Mesh and using an automatic meshing technique.

Converting GMS data to a 3D Mesh

3D Grids and 3D Scatter Points can be converted to a 3D Mesh. This is accomplished by using the following commands:

Grid → 3D Mesh

A 3D grid can be converted into a 3D mesh. If the 3D grid is a mesh-centered grid, the grid nodes are simply converted into a mesh nodes. If the 3D grid is a cell-centered grid, a mesh node is placed at the centroid of each cell to form the 3D mesh. An eight node quadrilateral element is created from each cell in the grid.

Scatter Points → Mesh Nodes

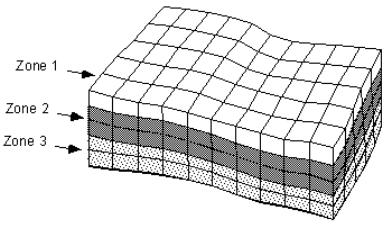
The ***Scatter Points → Mesh Nodes*** command is used to convert each of the scatter points to a 3D mesh node. The nodes can then be used to generate a mesh using the ***Tessellate*** command in the *Mesh* menu in the 3D Mesh module.

Automated Meshing

3D finite element meshes are not always constructed within the 3D Mesh module. The following methods are available for the construction of 3D Meshes:

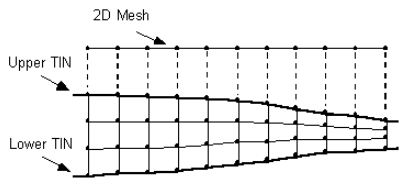
Fill Between TINs → 3D Mesh

3D meshes are often constructed using a combination of tools in the TIN module and the 2D Mesh module. Portions of the mesh corresponding to "zones" or stratigraphic units are constructed one at a time as shown below. Each of these zones is bounded above and below by a surface and consists of one or more layers of 3D elements.



3D Mesh With Multiple Zones

Before constructing a zone of elements, a 2D mesh must be created or imported using the 2D Mesh Module. A pair of TINs must also be created which represent the top and the bottom of the zone. These TINs are typically constructed from borehole data or from scatter points. The zone is then created by selecting the two TINs and selecting the ***Fill Between TINs → 3D Mesh*** command in the *TINs* menu. At this point, the user is prompted to enter the number of layers of elements to be created between the TINs and the material that will be associated with the elements in the zone. Each of the elements in the 2D mesh is then "projected" through the two TINs to create a vertical column of 3D elements as shown below. For example, if N layers are specified, N 3D wedge elements are created from each of the triangular elements in the 2D mesh, and N 3D hexahedral elements are created from each of the quadrilateral elements in the 2D mesh. The Z coordinates of the nodes created for the 3D elements are distributed uniformly between the top and the bottom TINs.



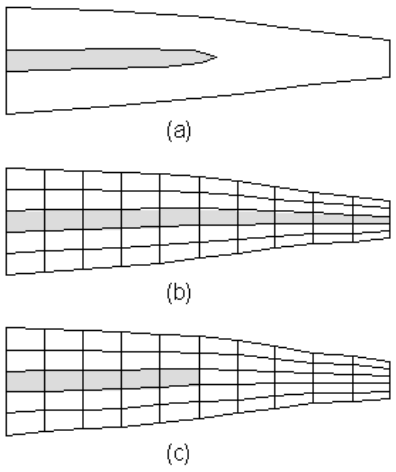
Projection Technique for Creating 3D Meshes

This process is repeated for each of the zones in the mesh. In order for the nodes at the bottom of one zone to match the nodes at the top of another zone, the same TIN should be used at the bottom of the upper zone and at the top of the lower zone. If the vertices of the TIN are edited in any way after one layer is generated but before an adjacent layer is generated, a gap may be introduced between the two zones of 3D elements.

Classify Elements

One way to model features such as a clay seam is to create all of the layers in the mesh and then change the material type of selected elements. The ***Classify Elements*** command in the *Mesh* menu can be used to accomplish the same task using solid models of the soil stratigraphy. Using this command, a solid model can be constructed and used to change the material type of a set of elements corresponding to a complicated geometric feature. When the ***Classify Elements*** command is selected, the centroid of each element in the 3D mesh is computed and the centroid is checked with each of the solid models to determine which solid the centroid lies within. The material type of the element is then changed to correspond to the material type of the solid containing the element centroid. If the centroid of an element does not lie in the interior of any of the solids, the material type of the element is unaltered.

The advantage of this construction procedure for 3D meshes is that it is simple and it is fast. The disadvantage of the procedure is that truncations or pinchout zones in the stratigraphy are not directly modeled. However, such features can be simulated by selecting elements and changing the material type associated with the elements once a zone of elements has been created. For example, suppose an aquifer contains a clay lens that extends partially into the aquifer as shown in part a of the figure below. A zone of elements could be created for the clay layer which extends over the entire XY range of the model (part b). The elements in this set of clay elements that are not in the region actually occupied by the clay layer could be selected and assigned the material type of the aquifer (part c). This can also be accomplished with a Solid Model and the ***Classify Elements*** command.



Modeling a Clay Seam Using Zones of Elements

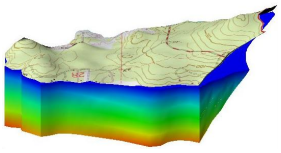
Creating 3D Meshes From Solid Models

Unlike the Solids → MODFLOW Boundary Matching option, it is not necessary to create a 3D mesh before selecting the ***Solids → Layered Mesh*** command. The 3D mesh is automatically generated. Each element in the 2D projection mesh is extruded into a vertical column of cells and the solids are used to assign the elevations to the nodes. The material ids are assigned to the 3D elements by finding the solid that encompasses the centroid of each 3D element assigning the material id of that solid to the element.

Creating 3D Meshes From Mesh Nodes

A mesh can be automatically constructed from a set of 3D nodes with the ***Tessellate*** command in the *Mesh* menu of the 3D Mesh module. This command performs the three-dimensional equivalent of the Delauney triangulation process. The Tesselation algorithm assumes that each of the vertices being tesselated is unique in xyz, i.e., no two points have the same xyz location. The result is a mesh composed entirely of tetrahedra. The region that is meshed corresponds to the convex hull of the nodes.

Editing a 3D Mesh

3D Mesh Module

3D Mesh
3D Element Types
Creating a 3D Mesh
Editing a 3D Mesh
Converting 3D Meshes to Other Data Types
More
3D Mesh Display Options
3D Mesh Tool Palette
ADH Display Options
Solids to Layered Mesh
3D Mesh Commands

Editing Nodes

3D mesh nodes can be:

- **Moved –** The coordinates of a 3D mesh node can be edited by selecting the node and dragging it to its new location or by typing the new coordinates in the Edit Window.
- **Locked –** Once a mesh has been created and edited as desired, the locations of all of the mesh nodes can be locked using the ***Lock All Nodes*** command. This is generally done to avoid inadvertent movement of the nodes while assigning boundary conditions and manipulating the view. Once the nodes can be unlocked by unselecting ***Lock All Nodes*** command.
- **Deleted –** 3D mesh nodes can be deleted by selecting the node and then the ***Delete*** command from the *Edit* menu or the ***Delete key***.
- **Renumbered –** As a 3D mesh is constructed within GMS, the nodes and elements in the mesh are numbered arbitrarily. If any nodes or elements are deleted, gaps are created in the numbering sequence. Such gaps can be removed and an optimal numbering sequence can be achieved by selecting the ***Renumber*** command in the *Mesh* menu. Prior to selecting the ***Renumber*** command, the user should select a series of boundary faces of the 3D mesh. These faces represent the location where the numbering process is to begin. In most cases, it is best to select all of the faces on an entire side of the mesh. This can be accomplished using the ***Select Face*** tool with the ***Control key*** held down. The renumbering process renumbers the nodes and elements in a logical order that tends to minimize the node and element bandwidth (which leads to more efficient solutions with some finite element solvers). The process begins by ordering the nodes and faces of the selected group of faces. This is essentially a 2D renumbering process. The longitudinal and lateral directions of the region of selected faces are determined and the numbering proceeds by sweeping along rows oriented in the lateral direction while progressing from row to row in the longitudinal direction. Once the nodes and faces of the selected region are renumbered, the layer of elements adjacent to the faces are numbered in a similar sequence. This process is repeated by sweeping outward

from the selected region, one layer of elements at a time, until the entire mesh is renumbered. The results of the renumbering process can be reviewed by turning on the display of node and/or element numbers in the Display Options dialog. The results can also be viewed by right-clicking on the 3D Mesh Data Folder in the Project Explorer and selecting the ***Properties*** command. The node and element bandwidths are listed in the dialog that comes up. If the objective of renumbering the mesh is minimizing the node and element bandwidths, the best results are generally achieved by selecting a side of the mesh corresponding to one of the two "ends" of the major or longitudinal axis of the mesh.

Refine Elements

3D mesh elements can be refined. Increasing the density of mesh elements can be accomplished by selecting a set of elements and selecting the ***Refine Elements*** command from the *Mesh* menu. This brings up the Refine Elements dialog.

Elements to Refine

The top portion of the dialog is used to specify which elements in the mesh are to be refined. If the Refine all 3D mesh elements option is selected, all elements in the mesh are refined regardless of which elements are selected. If the Refine selected 3D mesh elements option is selected, only the selected elements of the mesh are refined.

Even if the Refine selected 3D mesh elements option is selected, a few elements that were not selected must also be altered. This is due to the fact that the elements that were selected for refinement are refined, disjoint faces are created between the selected elements and the non-selected elements directly adjacent to the selected elements. To eliminate these disjoint faces, some transition elements are identified and refined. Transition elements are defined as any non selected element that shares at least one node with an element that is selected for refinement.

Refinement Method

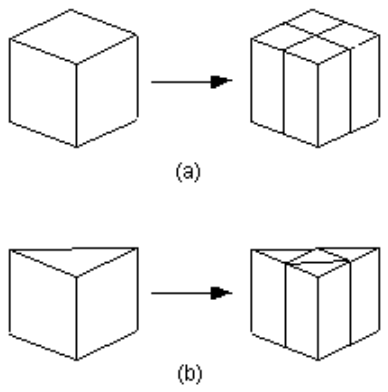
There are three methods of refinement that can be used. The difference among the three methods is the shape of the resulting mesh elements. Each of the three methods is described below.

Vertical Column

Vertical column refinement is used to split hexahedra and wedges in the X and Y directions only, as shown on the right.

Vertical column refinement was designed to be used with meshes created by extruding a 2D mesh through several layers. Meshes created in this manner are composed strictly of hexahedra and wedges and can be made by following the mesh extrusion procedure.

Depending upon the type and orientation of the elements in a 3D mesh, vertical column refinement may not be possible. When the *Refine Elements options* is selected from the *Mesh* menu, the entire mesh is checked to see if it can be refined using vertical column refinement. If vertical column refinement is not possible, the Vertical column refinement option is dimmed. In order for vertical column refinement to be possible, the following conditions must be met.



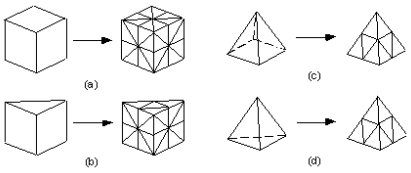
Vertical column refinement of (a) hexahedra and (b) wedges

1. If the entire mesh is to be refined, all elements in the mesh must be either hexahedra or wedges.
2. If only a selected portion of the mesh is to be refined, all selected elements must be either hexahedra or wedges.
3. All wedges to be refined must be oriented in space such that their top and bottom faces correspond to the triangular faces of the wedge.

4. Both the top and bottom faces of each element to be refined must be on the boundary or adjacent to other elements that are also to be refined.
5. All transition elements (i.e., elements not intended to be refined but share at least one node with an element that was selected for refinement) must also satisfy conditions 2 and 3 above.

All Elements To Tetrahedra

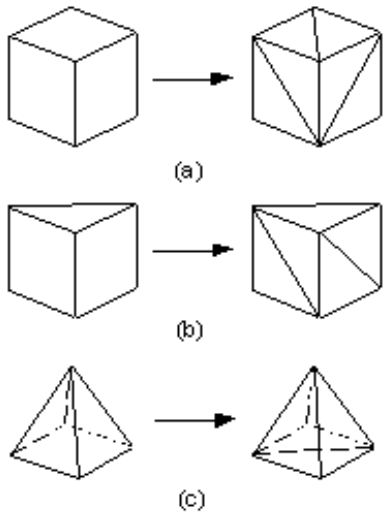
All element types to tets refinement is used to convert any of the four basic element types to tetrahedra. This option is especially useful since some finite element solvers require meshes to be composed strictly of tetrahedra.



All elements to tetrahedra fine method of refinement of (a) hexahedra, (b) wedges, (c) pyramids, and (d) tetrahedra

The Coarse refinement and Fine refinement options are used to specify the degree of refinement to be applied. If the Fine refinement option is selected, each tetrahedron is divided into eight smaller tetrahedra, each pyramid is divided into 16 smaller tetrahedra, each wedge is divided into 24 smaller tetrahedra, and each hexahedron is divided into 48 smaller tetrahedra. As with vertical column refinement, it is possible to refine either the entire mesh or selected portions of a mesh using the Fine refinement method.

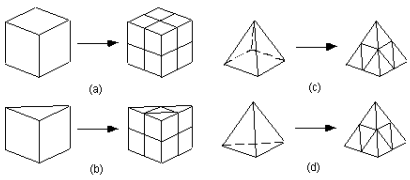
If the Coarse refinement option is selected, each pyramid is divided into two smaller tetrahedra, each wedge is divided into either three, or eight tetrahedra, and each hexahedra is divided into five, six, or twelve tetrahedra as shown below. Tetrahedra are not refined. Unlike the Fine refinement method, it is not possible to refine only a selected portion of a mesh when using the coarse method. The entire mesh gets refined.



All elements to tetrahedra coarse method of refinement of (a) hexahedra, (b) wedges, and (c) pyramids

Retain Element Types

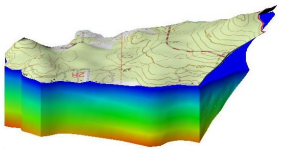
Retain element types refinement is used to convert any of the four basic element types to smaller elements of the same type. For example, each hexahedra is divided into eight smaller hexahedra as shown below. Pyramids are the exception since they are divided into five smaller pyramids and four tetrahedra. It is possible to divide a pyramid into four smaller pyramids, but the resulting pyramids are of poor quality.



Retain element types refinement of (a) hexahedra, (b) wedges, (c) pyramids, and (d) tetrahedra.

Like vertical column refinement, it is possible to refine only selected portions of a mesh when using Retain element types refinement. However, it is not always possible to retain element types in the transition elements. If the original mesh is composed of strictly tetrahedra, any selected region of the mesh can be refined without introducing elements other than tetrahedra. However, if the mesh contains any element type other than tetrahedra, pyramids and wedges will be introduced into the transition region.

3D Mesh Display Options

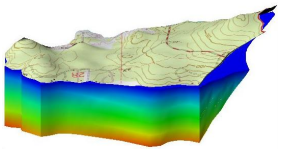
3D Mesh Module

3D Mesh
3D Element Types
Creating a 3D Mesh
Editing a 3D Mesh
Converting 3D Meshes to Other Data Types
More
3D Mesh Display Options
3D Mesh Tool Palette
ADH Display Options
Solids to Layered Mesh
3D Mesh Commands

The properties of all 3D Mesh data that GMS displays on the screen can be controlled through the 3D Mesh tab of the Display Options dialog. This dialog is opened by right clicking on the 3D Mesh Data entry in the Project Explorer and selecting the ***Display Option***s command. It can also be accessed from the from the *Display* menu or the Display Options macro. The following table describes the display options available for the 3D Mesh module.

Display Option	Description
Nodes	The Nodes item is used to display the mesh nodes.
Element edges	The Elements item is used to display the edges of elements. The elements are drawn using the color of the material associated with each cell. An option is included to display all of the edges or only the edges on the boundary of each material. Element can also be temporarily hidden. The color of the element edges can be adjusted according to the following options: 1. Auto – draws the material color if faces are not displayed. Uses black or white if the faces are displayed. 2. Specified – used the color specified next to the cell edges 3. Material – displays the material color of the cell
Element faces	This option fills the elements with the material color.
Texture map image	The Texture Map Image Item is used to "drape" an image over the top surface of the 3D Mesh.
Mesh shell	The Mesh shell item is used to display an edge for each of the edges on the exterior of the set of all elements (visible or invisible) which corresponds to a discontinuity in the mesh exterior. This display option provides a helpful spatial context when displaying iso-surfaces or cross sections.
Feature angle	The Mesh shell feature angle is used only when the Mesh Shell option is selected. This angle represents a threshold angle at which an edge of the shell will be displayed. If for example, an angle of 45 degrees is defined, any edge of the mesh which divides two element faces that are at an angle greater than 45 degrees to each other will not be displayed.
Node numbers	The Node numbers item is used to display the ID associated with each node next to the node. The numbers are only displayed on the front-facing faces of exterior elements.

Element numbers	The Element numbers item is used to display the ID associated with each element at the centroid of the element. The numbers are only displayed on the front-facing faces of exterior elements.
Scalar values	The Scalar Values item is used to display the scalar values of the active dataset for each node next to the node.
Contours	Most of the objects supported by GMS can be contoured by turning on the Contour Options in the Display Options dialog. When an object is contoured, the values associated with the active dataset for the object are used to generate the contours.
Vectors	If the Vectors item in the Display Options dialog is selected for an object (TIN, Grid, or Mesh), vector plots can be generated using the active vector dataset for the object. One vector is placed at each node, cell, or vertex.
Iso-surfaces	If the Iso-Surfaces item in the Display Options dialog is selected for an object (3D Grid or 3D Mesh), iso-surfaces will be generated. An iso-surface is the 3D equivalent of a contour line. While a contour line is a line of constant value extracted from a surface, an iso-surface is a surface of constant value extracted from a 3D dataset.

3D Mesh Tool Palette

3D Mesh Module

3D Mesh
3D Element Types
Creating a 3D Mesh
Editing a 3D Mesh
Converting 3D Meshes to Other Data Types
More
3D Mesh Display Options
3D Mesh Tool Palette
ADH Display Options
Solids to Layered Mesh
3D Mesh Commands

The following tools are contained in the dynamic portion of the Tool Palette when the 3D Mesh Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the 3D Mesh tool palette.

Tool	Tool Name	Description
	Select Boundary Nodes	The Select Boundary Nodes tool is similar in function to the Select Nodes tool except that it selects only nodes that are on the boundary of the mesh. This tool is useful when assigning nodal boundary conditions. All of the standard multi-selection techniques are available with this tool. In addition, if the Control key is depressed when a selection is made, all nodes on the same "side" of the mesh as the selected node are automatically selected. This option is useful when the same boundary condition is to be assigned to all nodes on the selected mesh side. The extent of the selected "side" is determined by feature breaks on the exterior of the mesh. If the angle between two adjacent element faces on the mesh is sharp, the common edge of the faces is assumed to be a feature break and is the boundary of a mesh side.
	Select Boundary Faces	The Select Boundary Faces tool is similar in function to the Select Boundary Nodes tool except that it selects faces of elements on the boundary of the mesh. This tool is useful when assigning flux type boundary conditions. All of the standard multi-selection techniques are available with this tool. In addition, if the Control key is depressed when a selection is made, all element faces on the same "side" of the mesh as the selected face are automatically selected. This option is useful when the same boundary condition is to be assigned to all faces on the selected mesh side. The extent of the selected "side" is determined by feature breaks on the exterior of the mesh. If the angle between two adjacent element faces on the mesh is sharp, the common edge of the faces is assumed to be a feature break and is the boundary of a mesh side.
	Select Material Zones	The Select Material Zones tool is used to select all elements of the mesh that have the same material type. This tool is useful for hiding or isolating zones in the mesh corresponding to a material type. When this tool is active, an icon appears on the mesh display for each of the material types. A material zone is selected by selecting the icon.

	Select Elements	The Select Elements tool is used to select individual elements. Elements are typically selected for hiding, or for changing the material type associated with the element. Multi-selection can be performed by holding down the Shift key while selecting or by dragging a rectangle to enclose the elements to be selected. The ID of the selected element is displayed in the Edit Window. Only visible elements can be selected. Elements which have been hidden cannot be selected. Hidden elements can be made visible by selecting the Show command in the Display menu. When selecting elements by dragging a box, all elements that lie within the box are selected. When selecting elements by clicking on individual elements with the cursor, only elements on the exterior of the visible portion of the mesh are selected. Elements in the interior of the mesh can be selected individually by first hiding the elements surrounding the elements to be selected. An element can also be selected by using the Find Element command in the Mesh menu user is prompted for an element ID and the element is selected. Any previously selected elements are unselected.
	Select Nodes	The Select Nodes tool is used to select individual nodes for editing. Multi-selection can be performed by holding down the Shift key while selecting or by dragging a rectangle to enclose the nodes to be selected. The ID of the selected node is displayed in the Edit Window. The coordinates of the selected node are also displayed in the Edit Window and can be edited by typing in new coordinates and selecting the TAB or Return key. Nodal coordinates can also be edited by dragging a node using the Select Nodes tool. When in plan view, nodes can be dragged in the XY plane. In any other view, nodes are constrained to move along the Z axis when they are being dragged. Since it is possible to accidentally drag points, nodes can be "locked" to prevent them from being dragged by selecting the Lock All Nodes command from the Mesh menu. The nodes can be unlocked by selecting Unlock All Nodes from the Mesh menu. A node can also be selected by using the Find Node command in the Mesh menu. The user is prompted for a node ID and the node is selected. Any previously selected nodes are unselected.
	Select Node Strings	The Select Node Strings tool is used to select one or more strings of nodes. Node strings are used for operations such as adding breaklines to the mesh. The procedure for selecting node strings is somewhat different than the normal selection procedure. Strings are selected as follows: • Click on the starting node for the string. The node selected will be highlighted in red. • Click on any subsequent nodes you would like to add to the string (nodes do not have to be adjacent). The selected nodes are now connected by a solid red line. To remove the last node from a string, press the Backspace key. To abort entering a node string, press the ESC key. To end a node string, press Return or double-click on the last node in the string. Another node string can then be selected.
	Select Wells	The Select Wells tool is used to select nodes which have a well (point source/sink) type boundary condition assigned to them. Since wells are often assigned to nodes in the interior of the mesh, it may be difficult to select the node that a well has been assigned to using the Select Nodes tool due to the large number of nodes in a mesh. This tool makes this type of selection easier since only well nodes can be selected when the tool is active.
	Select Cross Sections	Once a set of cross sections has been created, they can be selected using the Select Cross Sections tool. Selected cross sections can be deleted, or they can be made visible or invisible using the Hide and Show commands. When this tool is active, a cross section icon appears on each cross section. A cross section is selected by selecting the icon. When a different tool is selected, the icons disappear. When there are several cross sections, it is often easier to differentiate cross section icons in plan view (assuming the cross sections were created in plan view). As a general rule the icons are placed in the center of the first line segment used to cut the cross section.
	Select Particle Starting Locations	Particle Starting Locations, used in particle tracking, can be selected with the Select Particle Starting Locations tool. Selected particles can be deleted. Statistical information for the selected particles, such as the path length and time, is displayed in the status bar.
	Create Cross Section	Cross sections can be created from a 3D mesh using the Make Cross Section tool. Cross sections are formed when the user enters a polyline. A polyline is entered by clicking on several points and double-clicking on the final point when the line is finished. The Delete or Backspace key may be used to remove a point from the polyline, and the ESC key can be used to abort the process. A cross section or fence diagram is then computed by cutting perpendicular to the current viewing orientation through the currently visible elements of the mesh. While most cross sections are created with the mesh in plan view, any viewing orientation can be specified. Once cross sections are created, they can be deleted, hidden, or shown using the Select Cross Sections tool. Data sets are automatically interpolated from the 3D mesh to the cross sections for generation of contour and color fringe plots.

	Define Tetrahedron	Four tools are provided for interactively creating the four types of elements supported in GMS. While it is not practical to create an entire mesh with these tools, they are often useful for editing an existing mesh. The following steps are taken to construct individual elements: • Click on the first node. The node will be highlighted in red. • Click on the remaining nodes, one at a time, in the standard order for the element type. If the wrong node is selected, hitting the Delete or Backspace key backs the process up by one node. Hitting the ESCAPE key aborts the entire process.
	Define Pyramid Element	
	Define Wedge Element	
	Define Hexahedron Element	

Classify Material Zones

The classify material zones dialogs allows the user to assign materials to a grid or mesh. The source of the materials can be solids or a different grid or mesh.

Select background object

The background object defines the source of the materials to be assigned to the grid or mesh.

Classify algorithm

Two options are available to assign materials to the grid or mesh: Centroid and Predominant material.

When the Centroid method is used, the centroid of the grid cell or mesh element is calculated. Then the location of the calculated centroid is found in the background object (grid, mesh, or solid) and the material at that location in the background object is assigned to the grid cell or mesh element. If the calculated centroid is outside of the background object then the material of the grid cell or mesh element is not changed.

When the Predominant material method is used to assign materials to a grid from a solid the following process occurs. A vertical ray from the center of the cell is intersected with the Solids. The top and bottom of the grid cell is then compared with intersected solids to determine the length of each solid within the cell. Then all of the "solid lengths" with the same material id are combined. The material id with the greatest length in the cell is assigned to the cell. If more than one material has the same length in the cell then the material with the lowest id is assigned to the cell.

Material set name

The material set name is used to specify the name of the new material set as it appears in the project explorer.

Iso Surfaces

Iso-surface rendering is a powerful tool for visualizing 3D data sets. Iso-surfaces can be generated for 3D grids and 3D meshes. An iso-surface is the 3D equivalent of a contour line. While a contour line is a line of constant value extracted from a surface, an iso-surface is a surface of constant value extracted from a 3D dataset.

Defining Iso-values

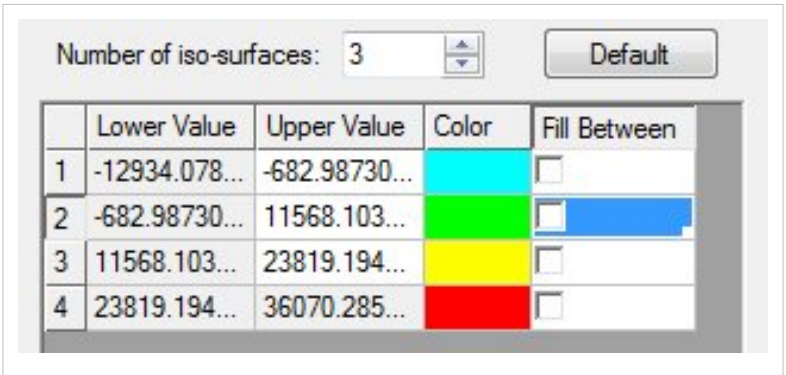
Iso-surfaces are computed using the active scalar data set for the grid or mesh. The Iso-surface Options dialog is accessed through the Iso-surface Options command in the Data menu or through a button in the 3D Grid Display Options or 3D Mesh Display Options dialogs. The items in the Iso-surface Options dialog are as follows:

Active Data Set

At the top of the dialog the active scalar dataset and active time step is listed. The maximum and minimum dataset values are also listed.

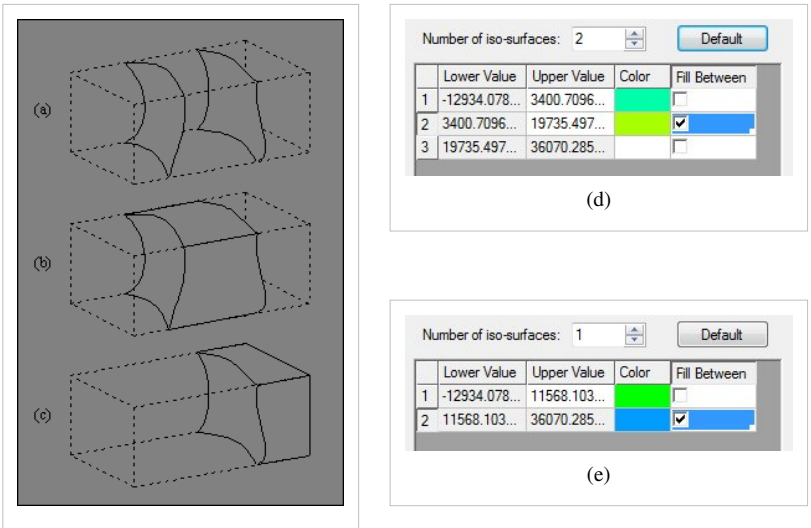
Iso-Values

In the next section of the dialog the number of iso-surfaces and the iso-values are defined. A maximum of 12 iso-surfaces may be created. The Default button can be used to automatically set up a number of iso-values. For example, if the number of iso-surfaces is three and the button is selected, three iso-values, equally spaced between the maximum and minimum dataset values are generated.



Fill Between

The Fill Between boxes to the right of the iso-values are used to generate surfaces on the exterior of the mesh or grid between two iso-values. For example, in part a of the figure shown below, two iso-surfaces have been generated using two iso-values. The image shown in part b was computed using the same iso-values and with the Fill Between box checked similar to figure d below. This causes the region of the mesh or grid boundary between the two iso-values to be defined as surfaces. The image shown in part c was generated using a single iso-value with the Fill Between box checked on the line after the iso-value like figure e. This causes the boundary with data set values greater than the specified value to be defined as surfaces.



Iso-Surfaces (a) No Fill (b) Fill Between Two Iso-Values (c) Fill Above or Below an Iso-Value (d) Filling between 2 iso-surfaces (e) Filling above an iso-surface.

Transparency

Transparency can be specified for each Iso-surface. Transparency affects the display of iso-surface faces.

Define as Cross-Section

Like contour lines, iso-surfaces are temporary in nature. In other words, if the active data set is changed, the current iso-surfaces are deleted and new iso-surfaces are computed using the new data set values. In some cases, it is useful to create an iso-surface as a permanent object. This can be accomplished by selecting the Define as cross section option in the Iso-surface Options dialog. This causes the computed iso-surfaces to be treated as cross sections. As cross sections, these iso-surfaces can be saved to a file, hidden, or deleted. In addition, if a new data set is selected, the iso-surfaces are not deleted. In fact, the values associated with the new data set are interpolated to the cross section iso-surfaces and can be displayed on the iso-surfaces as color fringes or contours. This makes it possible to effectively display two data sets at once.

Interior Edge Removal

By default, whenever an iso-surface is computed from a mesh or a grid, the lines corresponding to the intersection of the iso-surface with the cell or element boundaries are displayed on the iso-surfaces. If the Interior edge removal option is selected, only the edges on the iso-surface corresponding to a feature angle break greater than the specified value are displayed on the iso-surface. For example, if the feature angle were set at 30.00 degrees, the angle formed by the two polygonal faces adjacent to each edge in the iso-surface would be checked and only those edges where the computed angle is less than 180 - 30 = 150 degrees would be displayed. Typically, a small value (e.g., 0.001) is used so that only the edges adjacent to two coplanar faces are removed (made invisible).

Visible Region Only Option

If the Compute for visible region only option is chosen, the iso-surfaces are not computed in regions where the cells or elements are not visible. Otherwise, iso-surfaces are computed for all regions of the mesh or grid.

Iso-Surface Edges

If this option is on, the lines that make up the iso-surface are displayed.

Iso-Surface Faces

If this option is on, the iso-surface is displayed a color-filled surface.

Specify Range

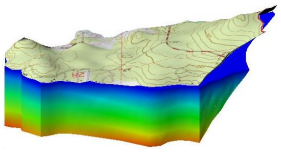
The specified range command allows the user to control the minimum and maximum value used to define the colors assigned to the iso-surfaces.

Isosurface Volumes

The volume within an iso-surface or the volume between two iso-surfaces is computed using the ***Iso-Surface Volume*** command in the *Data* menu. This command brings up a dialog with a list of iso-values and volumes. The listed iso-values correspond to the iso-values defined in the Iso-Surface Options dialog. The listed volumes represent the volumes between each of the iso-values. For example, the first volume represents the volume below (on the "low" side of) the lowest iso-value, the second volume represents the volume between the first and second iso-values, etc. The total volume listed at the bottom of the dialog should correspond to the total volume of the grid or mesh.

Beginning with GMS 7.0 the Isosurface Volumes dialog has changed to display a single volume associated with each isosurface. In addition to showing a single volume, the user can also view the volume of the cells (or elements) that are less than the isovalue, the volume of the cells that are greater than the isovalue, and the volume of the cells intersected by the isosurface by turning on the Display additional volume information toggle. In some cases GMS is unable to compute the volume of the isosurface. In those cases the user can still have an estimate of the isosurface volume by looking at the volume of the cells. The volume of the cells should be close to the volume of the isosurface when the grid (or mesh) is composed of very "small" cells. The difference between the isosurface volume and the volume of the cells decreases as the size of the cells decreases.

Converting 3D Meshes to Other Data Types

3D Mesh Module

3D Mesh
3D Element Types
Creating a 3D Mesh
Editing a 3D Mesh
Converting 3D Meshes to Other Data Types
More
3D Mesh Display Options
3D Mesh Tool Palette
ADH Display Options
Solids to Layered Mesh
3D Mesh Commands

3D Mesh Nodes can be converted to 3D Scatter Points bye using the following command in the *Mesh* Menu:

Mesh to 3D Scatter Points

The ***Mesh → Scatter Points*** command in the *Mesh* menu is used to create a new scatter point set using the nodes in a mesh. A copy is made of each of the datasets associated with the mesh and the datasets are associated with the new scatter point set.

This command is useful for comparing the solutions from two separate simulations from different meshes. For example, if two simulations have been performed with slightly different meshes (base vs. plan) it may be useful to generate iso-surfaces or a fringe plot showing the difference between the solutions. It is possible to generate a dataset representing the difference between two datasets using the data calculator. However, the two datasets must be associated with the same mesh before the data calculator can be used. The datasets from one of the meshes can be transferred to the other mesh as follows:

1. Load the first mesh and its dataset into memory.
2. Convert the mesh to a scatter point set using the ***Mesh → Scatter Points*** command.
3. Delete the first mesh by selecting the ***Delete All*** command from the *Edit* menu.
4. Load the second mesh and its dataset into memory.
5. Switch to the 3D Scatter Point module and select an interpolation scheme using the ***Interpolation Options*** command in the *Interpolation* menu.
6. Interpolate the dataset to the second mesh by selecting the to ***3D Mesh*** command from the *Interpolation* menu.

   At this point, both datasets will be associated with the second mesh and the Data Calculator can be used to compute the difference between the two datasets. This same sequence of steps can be used to interpolate a dataset from a 3D grid to a 3D mesh, or vice versa.

3D Mesh to 2D Mesh

This command creates a 2D mesh from the upward facing elements of the 3D mesh. The materials of the 3D elements are preserved on the 2D mesh. The datasets associated with the 3D mesh are NOT transferred to the 2D mesh.

Building the 3D Mesh from the FEMWATER Conceptual Model

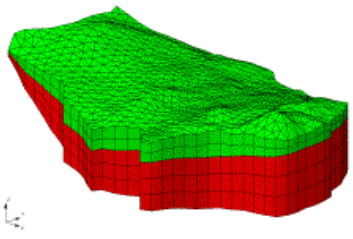
FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Once the FEMWATER conceptual model is constructed, the next step is to use the conceptual model to build a 3D finite element mesh. This is accomplished by first building a 2D mesh, then building the 3D mesh by extruding each of the 2D elements in 3D elements.

Map → 2D Mesh

The first step in building the 3D mesh is to select the ***Map → 2D Mesh*** command in the *Feature Objects* menu. This command creates a 2D mesh by automatically filling in the interior of the conceptual model with nodes and elements. The size and spacing of the elements is controlled by the spacing of the vertices on the arcs and by the refine point attribute assigned to any wells in the interior of the conceptual model.

An example of the ***Map → 2D Mesh*** command is shown in the following figure. A sample FEMWATER conceptual model is shown in part a. The 2D mesh resulting from execution of the ***Map → 2D Mesh*** command is shown in part b.



Converting a FEMWATER Conceptual Model (a) Conceptual Model (b) 2D Mesh Created with Map → 2D Mesh Command (c) 3D Mesh Created by Extruding 2D Mesh (d) 3D Mesh after ***Map → FEMWATER Command***.

Creating the 3D Elements

Once the 2D mesh is created, the next step is to create the 3D mesh by extruding each of the 2D elements into a series of 3D elements. The elevations of the 3D elements can be defined from a set of boreholes, a set of TINs, or a set of Solids.

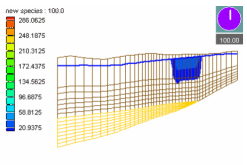
For sites with relatively simple stratigraphy, the ***Regions → 3D Mesh*** command in the Borehole module can be used.

For sites with more complex stratigraphy, the Fill Between ***TINs → 3D Mesh*** command in the TIN module should be used.

A mesh can also be created using the ***Solids → Layered Mesh*** command in the Solid module.

5.8. 3D Grid Module

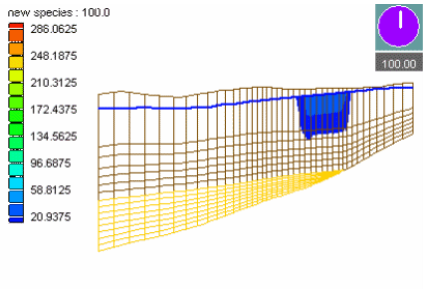
3D Grid Module

3D Grid Module

3D Grid
3D Grid Types
Creating and Editing
Viewing Modes
Converting to Other Data Types
Exporting Grids
More
Display Options
Tool Palette
Cell Properties
MODFLOW Display Options
MODPATH Display Options
MT3DMS Display Options
3D Grid Commands

The 3D Grid module is used to create 3D Cartesian grids. These grids can be used for interpolation, iso-surface rendering, cross sections, and finite difference modeling.

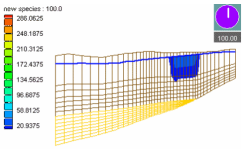
Interfaces to the following 3D finite difference models are provided in this module:

- MODFLOW
- MODPATH
- MT3DMS
- RT3D
- SEAM3D



Sample MT3D Simulation

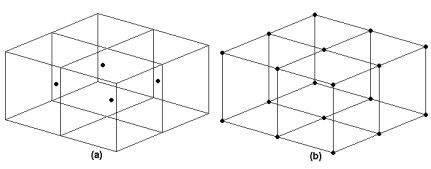
3D Grid Types

3D Grid Module

3D Grid
3D Grid Types
Creating and Editing
Viewing Modes
Converting to Other Data Types
Exporting Grids
More
Display Options
Tool Palette
Cell Properties
MODFLOW Display Options
MODPATH Display Options
MT3DMS Display Options
3D Grid Commands

Two types of 3D grids are supported in GMS: cell centered and mesh centered. When computations are performed on a mesh-centered grid, the computation points are the grid nodes or the corners of the grid cells. With a cell-centered grid, computations are performed at the cell centers.

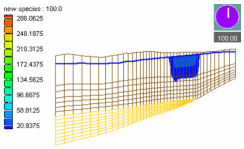
When a dataset is imported to a cell-centered grid, there is one value in the dataset for each cell. To use contouring or fringing the values at the cell corners must be known. Therefore, whenever contouring or fringing is performed, the values at the cell centers are interpolated to the cell corners. Interpolation to cell corners is only done for visualization. All computations performed using the Data Calculator are performed on the original values at the cell centers. With mesh-centered grids, all visualization and computations are performed at the cell corners and no interpolation is necessary.

All of the model interfaces in the 3D Grid module are based on cell-centered grids. Mesh-centered grids are useful for interpolation and iso-surface visualization since no extra interpolation is necessary.



(a) Cell Centered Grid (b) Mesh Centered Grid

Creating and Editing 3D Grids

3D Grid Module

3D Grid
3D Grid Types
Creating and Editing
Viewing Modes
Converting to Other Data Types
Exporting Grids
More
Display Options
Tool Palette
Cell Properties
MODFLOW Display Options
MODPATH Display Options
MT3DMS Display Options
3D Grid Commands

Creating 3D Grids

Two techniques are available for creating 3D grids: the ***Create Grid*** command in the 3D Grid Module and the ***Map → 3D Grid*** command in the Map Module. When a 3D Cell Centered Grid is created two different viewing modes are available.

Create Grid

A new grid can be created by selecting the ***Create Grid*** command from the *Grid* menu. This command brings up the Create Grid dialog. The options in the dialog are as follows:

Origin, Length, Rotation – By default, the rows and columns of 2D grids are aligned with the x and y axes. However, grids can be rotated about the z-axis, if desired. Thus, the information needed to determine the overall size and location of the grid is the xy coordinates of the lower left corner of the grid (the lower left corner prior to rotation), the length of the grid in the x and y directions, and the rotation angle. The xy coordinates of the origin are entered in the Origin edit fields, the dimensions are entered in the Length fields, and the angle of rotation is entered in the field entitled Rotation about Z-axis.

Bias – Several options are available for defining the number and locations of the cell boundaries. A bias can be defined which controls how the cell size varies from one cell to the next. For example, an X bias of 1.5 causes each cell to be 50% larger than the previous cell when moving in the positive x direction.

Number of Cells – The total number of cells in each direction (number of rows or columns) can be defined by explicitly entering a number or by entering a base cell size and a limit cell size. The base and limit cell size

options are used when a bias other than 1.0 is specified. The base cell size is the size of the first cell in the sequence. The cells are then generated by altering the cell size according to the bias until the limit cell size is reached. The remainder of the cells are constructed using the limit cell size.

Type and Orientation – The controls at the bottom of the Create Grid dialog are used to define the type and orientation of the grid. The user can specify whether the grid should be a mesh-centered grid or a cell-centered grid. The orientation of the ij axes with respect to the XY axes can also be specified.

Map → 3D Grid

Once the feature object coverages defining a conceptual model have been completely defined, the conceptual model is ready to be converted to a numerical model. The first step in this conversion process is to create a grid using the ***Map → 3D Grid*** command. Typically, the ***Grid Frame*** command is used prior to this command to define the location and dimensions of the grid.

When the ***Map → 3D Grid*** command is selected, the Create Grid dialog appears. If a grid frame has been defined, the size and location of the grid frame are used to initialize the fields in the Create Grid dialog. In most cases, these values will not need to be changed and the user can simply select the OK button to create the grid. If a grid frame has not been defined, the size and location of the grid are initialized so that the grid just surrounds the currently defined conceptual model. Once again, in most cases, no changes will need to be made and the user can typically immediately select the OK button to create the grid.

If one or more refine points are defined in the conceptual model, the number of rows and columns in the grid will be automatically determined when the grid is created. Thus, these fields cannot be edited by the user and will be dimmed. If refine points are not defined, the number of rows and columns must be entered.

When refine points are specified you must enter the Base size, Bias and Max size. The base size is the size you want the cell to be right at the refine point. The Max size is the largest size that you would like your cells to be in the entire grid. The bias determines how quickly the cell size will vary as you move away from the refine point. If you use a bias of 1.1 then the row next to the refine point will be 1.1 times the base size. The next row will be 1.1 size the previous row.

Editing 3D Grids

Each cell in a 3D grid has attributes associated with it. Each grid cell can be specified as active/inactive and each cell has a material associated with it. To edit the cell attributes associated with a numerical model see Cell Properties.

Rows, columns, or layers can be added or removed from a 3D grid. A row, column, or layer may be added to the grid by using one of the following tools found in the 3D Grid Tool Palette:

- Add i Boundary
- Add j Boundary
- Add k Boundary

The boundary of a cell can also be moved to a new location by using the Select Node tool. Existing rows, columns, or layers can be deleted by using the ***Merge Selected*** command in the Grid menu which is used to merge two or more selected rows, columns, or layers into a single row, column, or layer. Since the dimensions of the grid are changed, this command causes all datasets to be deleted. However, MODFLOW input parameters are preserved.

3D Grid Display Options

The properties of all 3D grid data that GMS displays on the screen can be controlled through the 3D Grid tab of the Display Options dialog. This dialog is opened by right clicking on the 3D Grid Data entry in the Project Explorer and selecting the ***Display Options*** command. It can also be accessed from the from the *Display* menu or the Display Options macro. The following table describes the display options available for the 3D Grid module.

Display Option	Description
Nodes	The Nodes item is used to display grid nodes depending on the Grid Type . If the grid is cell-centered, a dot is displayed at the cell centers. If the grid is mesh-centered, a dot is displayed on the cell corners.
Cell edges	The Cell edges item is used to display the edges of grid cells. The cells are either drawn using the default cell color or the color of the material associated with each cell. In addition to turning the display of cells on or off, you can temporarily hide grid cells. The color of the cell edges can be adjusted according to the following options: 1. Auto – draws the material color if faces are not displayed. Uses black or white if the faces are displayed 2. Specified – used the color specified next to the cell edges 3. Material – displays the material color of the cell
Cell faces	The Cell faces item causes the faces of the grid cells to be drawn as filled polygons.
Inactive cells	The Inactive cells item is used to display cells which are inactive. If this option is turned off, inactive cells are not displayed. Inactive cells must be displayed before they can be selected.
Named layer ranges legend	A legend showing the material and named layer ranges can be turned on.
Cell numbers	The Cell Numbers item is used to display the ID of each grid cell.
Node numbers	The Node Numbers item is used to display the ID of each grid node.
Scalar values	The Scalar Values item is used to display the scalar values of the active dataset for each node next to the node.
IJK indices	The IJK indices item is used to display the ijk indices of each cell or node.
IJK triad	The IJK triad item is used to display a symbol at one of the corners of the grid showing the orientation of the ijk axes.
True layer display	With MODFLOW models, a special option called the True Layer mode is available. If this mode is selected, the user provides a set of top and bottom elevation arrays for each layer. These arrays can be used to display the vertical variations in the stratigraphy when in one of the side views in orthogonal viewing mode or when in oblique view in general mode.
Layer borders	The Grid boundary item is used to display a solid line around the perimeter of the grid layers.
Grid shell	The Grid shell item is used to display a solid cube around the extents of the grid. Displaying the boundary is useful when iso-surfaces are being displayed with the cell edges turned off.
Texture map image	The Texture Map Image Item is used to "drape" an image over the surface of the 3D Grid.
Contours	Most of the objects supported by GMS can be contoured by turning on the Contour Options in the Display Options dialog. When an object is contoured, the values associated with the active data set for the object are used to generate the contours.
Specified Dataset	Allows the user to display the contours of a second dataset that is specified by the user. All contouring options are the same for both the specified and active datasets.
Vectors	If the Vectors item in the Display Options dialog is selected for an object (TIN, Grid, or Mesh), vector plots can be generated using the active vector data set for the object. One vector is placed at each node, cell, or vertex.
Iso-surfaces	If the Iso-Surfaces item in the Display Options dialog is selected for an object (3D Grid or 3D Mesh), iso-surfaces will be generated. An iso-surface is the 3D equivalent of a contour line. While a contour line is a line of constant value extracted from a surface, an iso-surface is a surface of constant value extracted from a 3D dataset.

3D Grid Tool Palette

The following tools are contained in the dynamic portion of the Tool Palette when the 3D Grid Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the 3D Grid tool palette.

Tool	Tool Name	Description
	Select Cells	The Select Cells tool is used to select individual grid cells. Multi-selection can be performed by holding down the Shift key while selecting or by dragging a rectangle to enclose the cells to be selected. The ijk indices of the selected cell are displayed in the Edit Window. Only visible cells can be selected. Cells which have been hidden cannot be selected. Inactive cells can only be selected when they are being displayed by turning on the Inactive Cells item in the Display Options dialog. When selecting cells by dragging a box, all cells that lie within the box are selected. When selecting cells by clicking on individual cells with the cursor, only cells on the exterior of the visible portion of the grid are selected. Cells in the interior of the grid can be selected individually by first hiding the layers, rows, or columns adjacent to the cells. Cells can also be selected by using the Find Cell command in the Grid menu. is used for locating cells in the current grid based on the ijk position within the grid or by cell ID. The Find Cell dialog provides edit fields for both an ID and an ijk value. Entering a value for ID will automatically update the ijk fields. Likewise, entering a value for the ijk location will automatically update the ID. When the OK button is selected, the indicated cell will be selected in the finite difference grid.In addition to selecting one cell at a time, the Find Cell Dialog can select an entire row column or layer. A zero may be entered in any of the i, j or k fields indicating that all cells in that direction will be selected. The ID of the cells that will be selected is also displayed as static text at the top of the dialog.
	Select Node	The Select Node tool is used select nodes and to interactively edit cell boundary coordinates by clicking on the intersection of two cell boundaries and dragging the boundaries with the mouse button held down. The coordinates of the cell boundary intersection are displayed in the Edit Window as the boundaries are dragged. The coordinates of a selected boundary intersection can also be edited by directly entering the coordinates in the Edit Window. When dragging a boundary intersection, the intersection is moved in the plane of the face where the point was clicked. For example, when a boundary intersection on the top of the grid is dragged, the intersection is constrained to move in the XY plane. If a boundary intersection on the side of the mesh perpendicular to the X axis is dragged, the intersection is constrained to move in the YZ plane. If the Control key is depressed when dragging a boundary intersection in a view other than plan view, the intersection is constrained to move in a plane parallel to the viewing plane.
	Select Material Zones	The Select Material Zones tool is used to select all cells of the grid that have the same material type. This tool is useful for hiding or isolating zones in the grid corresponding to a material type. When this tool is active, an icon appears on the grid display for each of the material types. A material zone is selected by selecting the icon.
	Select i	The Select i tool is used to select an entire "row" (set of cells with the same i index) of cells at once. Multi-selection can be performed by holding down the Shift key while selecting. The i index of the selected row is displayed in the Edit Window.
	Select j	The Select j tool is used to select an entire "column" (set of cells with the same j index) of cells at once. Multi-selection can be performed by holding down the Shift key while selecting. The j index of the selected column is displayed in the Edit Window.
	Select k	The Select k tool is used to select an entire "layer" (set of cells with the same k index) of cells at once. Multi-selection can be performed by holding down the Shift key while selecting. The k index of the selected layer is displayed in the Edit Window.
	Select Cross Sections	Once a set of cross sections has been created, each cross section can be selected using the Select Cross Sections tool. Selected cross sections can be deleted, or they can be made visible or invisible using the Hide and Show commands. When this tool is active, a cross section icon appears on each cross section. A cross section is selected by selecting the icon. When a different tool is selected, the icons disappear. When there are several cross sections, it is often easier to differentiate cross section icons in plan view (assuming the cross sections were created in plan view). As a general rule the icons are placed in the center of the first line segment used to cut the cross section.
	Select Particle Starting Locations	Particle Starting Locations, used in particle tracking (MODPATH), can be selected with the Select Particle Starting Locations tool. Selected particles can be deleted. Statistical information for the selected particles, such as the path length and time, is displayed in the status bar.

	Add i Boundary	The Add i Boundary tool is used to insert a new i boundary into the grid. The new boundary is inserted at the cursor location when the mouse button is clicked. Inserting a new cell boundary changes the dimensions of the grid and all datasets associated with the grid are deleted. If the control key is held down while executing this command, the row will be evenly divided.
	Add j Boundary	The Add j Boundary tool is used to insert a new j boundary into the grid. The new boundary is inserted at the cursor location when the mouse button is clicked. Inserting a new cell boundary changes the dimensions of the grid and all datasets associated with the grid are deleted. If the control key is held down while executing this command, the column will be evenly divided.
	Add k Boundary	The Add k Boundary tool is used to insert a new k boundary into the grid. The new boundary is inserted at the cursor location when the mouse button is clicked. Inserting a new cell boundary changes the dimensions of the grid and all datasets associated with the grid are deleted. If the control key is held down while executing this command, the layer will be evenly divided.
	Create Cross Section	The Create Cross Section tool is used to create cross sections in a 3D. Cross sections are formed when the user enters a polyline. A polyline is entered by clicking on several points and double-clicking on the final point when the line is finished. The Delete or Backspace key may be used to remove a point from the polyline, and the ESC key can be used to abort the process. A cross section or fence diagram is then computed by cutting perpendicular to the current viewing orientation through the currently visible cells of the grid. While most cross sections are created with the grid in plan view, any viewing orientation can be specified. Datasets are automatically interpolated from the 3D grid to the cross sections for generation of contour and color fringe plots.

Classify Material Zones

The classify material zones dialogs allows the user to assign materials to a grid or mesh. The source of the materials can be solids or a different grid or mesh.

Select background object

The background object defines the source of the materials to be assigned to the grid or mesh.

Classify algorithm

Two options are available to assign materials to the grid or mesh: Centroid and Predominant material.

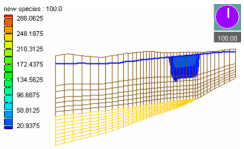
When the Centroid method is used, the centroid of the grid cell or mesh element is calculated. Then the location of the calculated centroid is found in the background object (grid, mesh, or solid) and the material at that location in the background object is assigned to the grid cell or mesh element. If the calculated centroid is outside of the background object then the material of the grid cell or mesh element is not changed.

When the Predominant material method is used to assign materials to a grid from a solid the following process occurs. A vertical ray from the center of the cell is intersected with the Solids. The top and bottom of the grid cell is then compared with intersected solids to determine the length of each solid within the cell. Then all of the "solid lengths" with the same material id are combined. The material id with the greatest length in the cell is assigned to the cell. If more than one material has the same length in the cell then the material with the lowest id is assigned to the cell.

Material set name

The material set name is used to specify the name of the new material set as it appears in the project explorer.

3D Grid Viewing Modes

3D Grid Module

3D Grid
3D Grid Types
Creating and Editing
Viewing Modes
Converting to Other Data Types
Exporting Grids
More
Display Options
Tool Palette
Cell Properties
MODFLOW Display Options
MODPATH Display Options
MT3DMS Display Options
3D Grid Commands

When a 3D cell-centered grid is in memory, two viewing modes are available: General Mode and Orthogonal Mode. The general mode is the default mode and it is the mode used when a cell-centered 3D grid is not in memory. In general mode you can view the grid from top, front, or side view or from any oblique view. With the orthogonal view, the viewing direction are restricted to three views: looking down one of the i, j, or k axes. As you look down an axis, you view one row, column, or layer at a time. Oblique views and shading are not available in the orthogonal mode. The default viewing mode for cell-centered 3D grids is the orthogonal mode. Thus, whenever a new cell-centered grid is created or read from a file, GMS automatically goes into the orthogonal viewing mode

There are two main advantages of the orthogonal mode:

- It is a convenient way to view and manipulate layered models such as MODFLOW.
- Since you only view one row, column, or layer at a time, there are fewer things to display. Thus, redrawing a grid is much faster.

Switching Modes

A command is provided in the *View* menu for switching between the orthogonal and general viewing modes. If the current mode is orthogonal, the menu command is titled ***Ortho Mode*** will be selected. If the current mode is general, the command is titled ***General Mode*** will be selected. There is also a toolbar Macro that can be used to switch the mode .

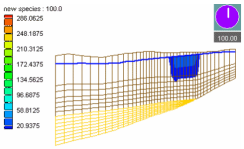
Mini-Grid Plot

When in the orthogonal mode, the **Mini-Grid Plot** is activated in the Tool Palette. The plot shows which row, column, or layer is currently being displayed. The edit field and arrows just beneath the plot can be used to change the current row, column, or layer. To change the view, select one of the View Along I Axis, View Along J Axis, or View Along K Axis macros at the bottom of the Tool Palette.

True Layer Mode

With MODFLOW models, a special option called the True Layer mode is available. If this mode is selected, the user provides a set of top and bottom elevation arrays for each layer. These arrays can be used to display the vertical variations in the stratigraphy when in one of the side views in orthogonal viewing mode or when in oblique view in general mode.

Converting 3D Grids to Other Data Types

3D Grid Module

3D Grid
3D Grid Types
Creating and Editing
Viewing Modes
Converting to Other Data Types
Exporting Grids
More
Display Options
Tool Palette
Cell Properties
MODFLOW Display Options
MODPATH Display Options
MT3DMS Display Options
3D Grid Commands

3D Grid data can be converted to other types of data in GMS such as 2D grids, 2D scatter points, 3D Meshes, or 3D scatter points. 3D Grid data is converted using the following commands in the Grid menu:

Grid → 3D Scatter Points

The ***Grid → Scatter Points*** command in the *Grid* menu in the 3D Grid Module is used to create a new scatter point set using the nodes or cells of a 3D grid. A copy is made of each of the datasets associated with the grid and the datasets are associated with the new scatter point set.

Grid → 3D Mesh

A new 3D finite element mesh can be created from a 3D grid by selecting the ***Grid → Mesh*** command from the *Grid* menu in the 3D Grid menu. An eight node quadrilateral element is created from each cell in the grid.

Grid → 2D Grid

A new 2D grid can be created from a 3D grid by selecting the ***Grid → 2D Grid*** command from the *Grid* menu in the 3D Grid module. This creates a 2D grid which matches the 3D grid, i.e., one cell is created in the 2D grid for each vertical (ij) column in the 3D grid. This command is typically used in conjunction with the ***3D Data → 2D Data*** command.

MODFLOW Layers → 2D Scatter Points

The ***MODFLOW Layers → 2D Scatter Points*** command in the *Grid* menu of the 3D Grid module is used for regional to local model conversion. It is only available if the true layer mode is being used with a MODFLOW model. When this command is selected, a new 2D scatter point set is created and a scatter point is created at the centroid of each vertical column of cells in the 3D grid. A dataset is then created on the scatter point set for the top and bottom elevations of each layer and for the computed head values (if a MODFLOW solution is in memory). The MODFLOW head data set is chosen in the following way: If a modflow head data set is the active dataset then GMS uses that dataset for the starting heads. If the active dataset is in a modflow solution but is not a head data set then GMS tries to use the head dataset in the active solution. If neither of the first 2 cases works then GMS just goes through the list of datasets and uses the last modflow head dataset that it finds (most likely the last solution that was read in). At a later point in time, these data sets can be interpolated from the scatter points to the cell centers of a smaller, local grid.

This dialog allows the user to create a scatter point set and data sets of the current MODFLOW simulation.

The user can limit the number of scatter points created by turning on the **Only create scatter points within selected coverage option** and selecting the appropriate coverage. This is often used when converting between a regional model and a local scale model.

- Layer Subdivision

  The layers of the current MODFLOW simulation can be subdivided by specifying the **Number of local model layers** for each layer in the current 3D grid.

  The user can select which MODFLOW datasets to create by turning off/on the toggles below the **Create datasets of** text. Datasets for layer elevations, flow package data (HK, HANI, VK, SY, SS...), Recharge, and Heads can be created. The user must select a 3D grid dataset in order to create a dataset for the Heads (most often this will be the MODFLOW solution from the regional model).

  The Recharge and Head datasets can be transient if the regional model was transient. The user can also select a Start and End time to limit the number of time steps for these datasets.

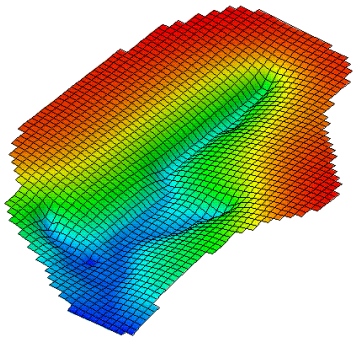
- Bias Layer 1

  An option to bias the thickness of the new layer 1 is also available. This can be useful if equally subdividing layer 1 of the regional model would result in cells where the elevation would be above the computed heads. Thus, the user can choose to bias the thickness of the new layer 1 to ensure that the top layer in the new model will not be dry.

3D Data → 2D Data

The ***3D Data → 2D Data*** command in the *Data* menu of the 3D Grid module is used to create datasets on a 2D grid created using the Grid to 2D Grid command. These two commands are useful for creating a 2D representation of a 3D dataset for contouring.

The ***3D Data → 2D Data*** command brings up the *3D Dataset → 2D Dataset* dialog. The button at the top of the dialog is used to select which 3D dataset is to be converted to a 2D dataset. The drop down box lists each of the options available for converting each column of 3D data values to a single 2D data value. The figure below shows an example of using the ***3D Data → 2D Data*** command.



Watertable Elevations From a MODFLOW Simulation

Exporting Grids

2D and 3D grids can be exported from GMS in various formats by right-clicking on the grid in the Project Explorer and selecting the *Export* command. The options available include:

Exporting 2D grids

1. Text GMS 2D Grid File (*.2dg)

   This is an older GMS grid file format that is described in this document [1].
2. ARC/INFO Ascii Grid File (*.asc)

   This is a raster format that can be opened in ArcMap. Because it's a raster, you can only use this option with cell-centered grids where all cells are the same size.

Exporting 3D grids

1. Text GMS 3D Grid File (*.3dg)

   This is an older GMS grid file format that is described in this document [1].
2. Shapefile

   An ESRI ArcGIS compatible shapefile will be exported containing 2D polygons for all 3D grid cells in all grid layers. The attribute table fields include:

   - cell ID
   - cell I
   - cell J
   - cell K
   - grid cell activity

- active scalar dataset values
- active dataset activity (if it exists)

A simple definition query using the grid cell activity and K fields can be created in ArcMap to see one layer of the grid at a time.

References

[1] http://gmsdocs.aquaveo.com.s3.amazonaws.com/fform30.pdf

Cell Properties

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The cell properties dialog allows the user to edit cell properties. Most cell properties are associated with a model such as MODFLOW or MT3D. If no models exist in the GMS project them the cell properties dialog will only allow editing of the material assigned to the grid cell.

MODFLOW

Several input arrays defining parameters such as starting head, IBOUND, hydraulic conductivity, and transmissivity are defined in the Global/Basic and BCF, LPF, or HUF packages. These arrays can be edited in the Basic and BCF/LPF/HUF Package dialogs, or they can be initialized using a conceptual model in the Map module. In many cases however, it is necessary to view or edit the values on a cell-by-cell basis. This can be accomplished using the ***Cell Properties*** command in the *MODFLOW* menu.

Before selecting the ***Cell Properties*** command, a set of cells should be selected using the cell selection tools. Once the command is selected, the *MODFLOW Cell Attributes* dialog appears.

The parameters for the selected cells are changed by typing in new values in the edit fields. If more than one cell is selected when the ***Cell Properties*** command is selected, the available edit fields will be left blank (unless all values are the same for that parameter). To edit one of the parameters, click on the edit field to be edited. When the **OK**

button is selected, only the parameters whose edit fields that have data are changed. This makes it possible to change one of the parameters (e.g., transmissivity) for all of the selected cells while leaving the other parameters unchanged.

NOTE: When you are using materials to define the MODFLOW model, the Cell Properties dialog will show the material properties relating to the material of the selected cell. You will not be able to edit these values on a cell-by-cell basis, but you can either edit the material type for this cell if the active material set is the default material set, or you can change the material properties for the material (which affects every cell that uses that material).

MT3DMS/RT3D/SEAM3D

MT3D inputs that vary on a cell by cell basis can also be editing using this dialog and editing the data in the MT3D tab.

Active/Inactive Cells

Each of the cells in a cell-centered grid can be active or inactive. An inactive cell is a cell that is not part of the computational domain. An inactive cell is ignored when contours or vectors are displayed on the grid. Several methods are available for changing the active/inactive status of cells.

- IBOUND/ICBUND Arrays

  The active/inactive status of cells can also be controlled with model parameters. For example, MODFLOW uses an array of values known as the IBOUND array, to indicate what is active and what is inactive. If dataset flags are not currently present in the active dataset, and a MODFLOW simulation is currently in memory, the active/inactive status of cells will be determined by the IBOUND array. The ICBUND array in MT3DMS also has an effect on the active/inactive flags.

- Activate Polygon Region

  Active/inactive status of cells can be set using the ***Activate Cells in Coverage*** command in the Map module. This command checks each cell in the grid to see if it is within the polygons defined in the MODFLOW/MT3DMS local Source/sink type coverage. All cells within the coverage are made active and all cells outside the coverage are made inactive. This command just modifies the IBOUND array in MODFLOW. If there is no MODFLOW model in memory, the command can hide the grid cells instead of making them inactive.

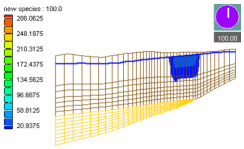
- Dataset Flags

  Often, the status of the cells of a finite difference grid will be determined from the solution to a numerical analysis. For example, a cell may go dry during a MODFLOW simulation, making the cell inactive. Two types of solution files supported by GMS may include active/inactive flags: GMS dataset files and MODFLOW solution files. After importing such a dataset, the active/inactive flags are stored with the datasets (or with the time steps of a transient dataset). When a dataset is selected as the active dataset, the flags (if they exist) are checked and any cell which is inactive is ignored when contouring and fringing. Active/inactive flags associated with datasets take precedence over any other method of specifying active/inactive status. When the dataset is switched or deleted, the active/inactive flags for the grid revert to their previous values.

- Active/Inactive Flags Dialog

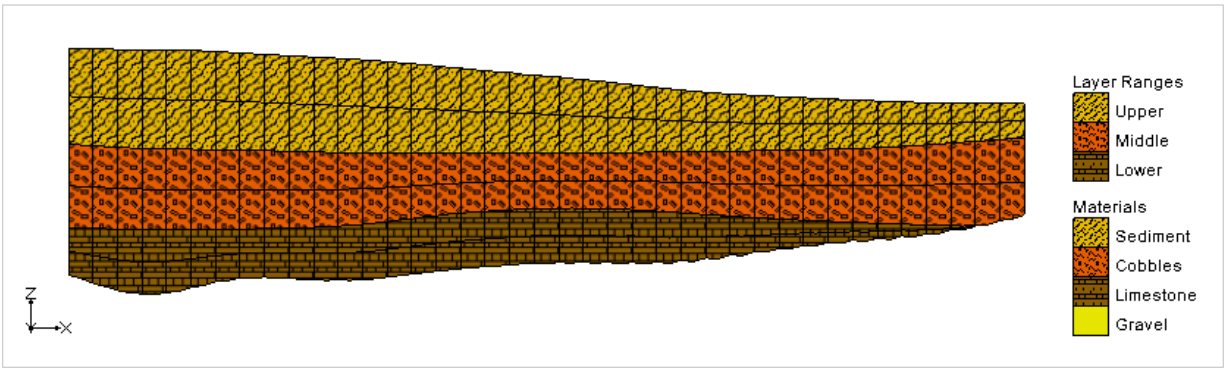
  In some cases, a dataset may not explicitly contain active/inactive flags, but the flags can be inferred from one or more key values. For example, a value of -999 in a dataset may mean that the cell is dry or inactive. A set of key values can be defined to set up the active/inactive flags for a dataset using the *Active/Inactive Flags* dialog. The *Active/Inactive Flags* dialog is accessed in the Dataset Info Dialog. The active/inactive status of the cells is determined from the specified key values in the list. Any number of key values may be specified.

Named Layer Ranges

3D Grid Module

3D Grid
3D Grid Types
Creating and Editing
Viewing Modes
Converting to Other Data Types
Exporting Grids
More
Display Options
Tool Palette
Cell Properties
MODFLOW Display Options
MODPATH Display Options
MT3DMS Display Options
3D Grid Commands

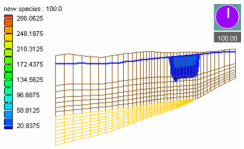
Starting at version 8.0, GMS allows you to create named layer ranges via the ***Grid|Named Layer Ranges*** menu command. A layer range has a name, a material, and a minimum and maximum layer. If named layer ranges are defined, GMS will create a material set called "Named Layer Ranges" that matches the ranges defined in the dialog (if the material set already exists it is simply updated). If the **Update grid on OK** toggle is on, GMS will make the "Named Layer Ranges" material set the active set. You can turn on a legend showing the named layer ranges.

Named layer ranges can be used in T-PROGS to target a subset of grid layers.



Redistribute Grid Cells

3D Grid Module



3D Grid

3D Grid Types

Creating and Editing

Viewing Modes

Converting to Other Data Types

Exporting Grids

More

Display Options

Tool Palette

Cell Properties

MODFLOW Display Options

MODPATH Display Options

MT3DMS Display Options

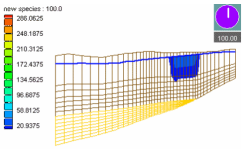
3D Grid Commands

The number of 3D grid rows, columns or layers can be changed via the *Redistribute Grid Cells* dialog. This dialog is accessed by selecting rows, columns or layers using the Select i, Select j or Select k tools, right-clicking and clicking the ***Redistribute*** menu command. The Redistribute Grid Cells dialog indicates the number of ranks (rows, columns or layers) that are selected and allows you to enter a new number. Entering a new number will cause GMS to increase or decrease the number of ranks. The new ranks are distributed evenly in the selected area and any model boundary conditions are positioned as close to their old locations as possible.

Other ways to alter grid rows, columns and layers:

- Grid ranks can be inserted manually using the Add i Boundary, Add j Boundary and Add k Boundary tools.
- Grid ranks can be merged manually using the Select i, Select j or Select k tools, right-clicking and selecting the Merge command.
- ***Redistribute Layers*** menu command.

Redistribute Layers

3D Grid Module

3D Grid
3D Grid Types
Creating and Editing
Viewing Modes
Converting to Other Data Types
Exporting Grids
More
Display Options
Tool Palette
Cell Properties
MODFLOW Display Options
MODPATH Display Options
MT3DMS Display Options
3D Grid Commands

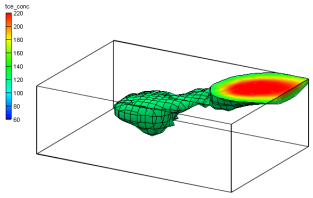
One way to alter 3D grid layer thicknesses is via the Redistribute Layers dialog accessed via the ***Grid\Redistribute Layers*** menu command. This dialog allows you to specify a constant elevation for the top and bottom of the grid and edit the relative thicknesses of the grid layers.

Other ways to alter grid layer thicknesses:

- dragging with the *Select Node* tool
- ***Redistribute Grid Cells*** popup menu command.

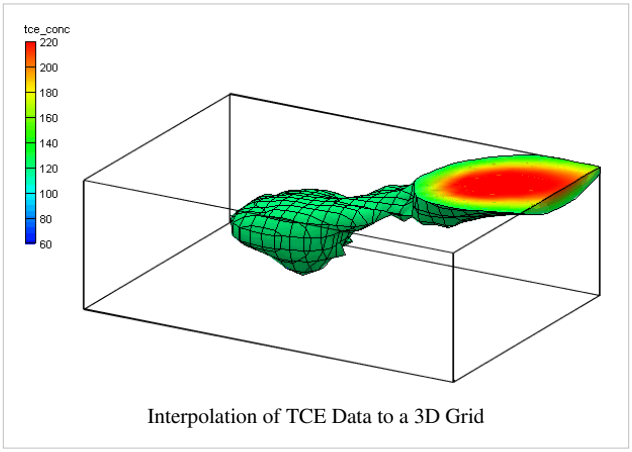
5.9. 3D Scatter Point Module

3D Scatter Point Module

3D Scatter Point Module

3D Scatter Point
Interpolating with 3D Scatter Points
Converting 3D Scatter Points to Other Data Types
Bounding Grid
More
3D Scatter Point Display Options
3D Scatter Point Tool Palette
Active/Inactive Points
Inverse Distance Weighted
3D Scatter Point Commands

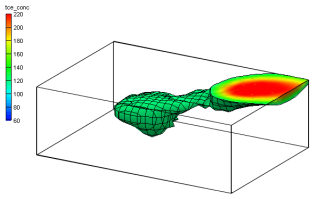
The 3D Scatter Point module is used to interpolate from groups of 3D scatter points to meshes, grids, or TINs. Several interpolation schemes are supported including kriging.

Interpolation is useful for setting up input data for analysis codes and it is also useful for site characterization.



Interpolation of TCE Data to a 3D Grid

3D Scatter Point Display Options

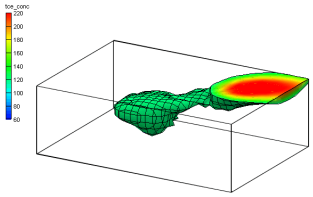
3D Scatter Point Module

3D Scatter Point
Interpolating with 3D Scatter Points
Converting 3D Scatter Points to Other Data Types
Bounding Grid
More
3D Scatter Point Display Options
3D Scatter Point Tool Palette
Active/Inactive Points
Inverse Distance Weighted
3D Scatter Point Commands

The properties of all 3D scatter data that GMS displays on the screen can be controlled through the 3D Scatter tab of the *Display Options* dialog. This dialog is opened by right clicking on the 3D Scatter Data entry in the Project Explorer and selecting the ***Display Options*** command. It can also be accessed from the from the *Display* menu or the Display Options macro. The following table describes the display options available for the 3D Scatter Point module.

Display Option	Description
Active scatter point set	The name of the active scatter point set is listed at the top of the dialog. The symbol selected using the Scatter point symbols option (described below) applies to the active scatter point set. This makes it possible to use a different set of symbols for the points in each set so that the sets are easily distinguishable.
Scatter point symbols	The Scatter point symbols item is used to display a symbol at the location of each scatter point. The widget to the left of the toggle is used to bring up a dialog listing the available symbols. The color of each of the scatter points in a set may be changed in this dialog also. The color of the scatter points can be adjusted according to the following options: 1. Specified – used the color specified next to the scatter point symbols 2. Data – the color ramp is used to assign a color to each of the symbols according to the value of the active scalar dataset
Inactive scatter points	Individual scatter points can either be active or inactive. The Inactive scatter points option is used to show inactive scatter points and to set their color.
Scatter point scalar values	The Scatter point scalar values option is used to display the value of the active dataset next to each of the scatter points.
Scatter point labels	The Scatter point labels item is used to display the scatter point label next to each scatter point.
Scatter point numbers	The Scatter point IDs item is used to display the scatter point ID next to each scatter point.

Symbol legend	The Symbol legend item is used to display a symbol legend listing each of the scatter point sets by name and showing the symbol associated with the scatter point sets.

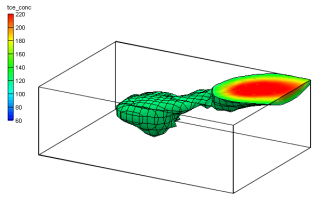
3D Scatter Point Tool Palette

3D Scatter Point Module

3D Scatter Point
Interpolating with 3D Scatter Points
Converting 3D Scatter Points to Other Data Types
Bounding Grid
More
3D Scatter Point Display Options
3D Scatter Point Tool Palette
Active/Inactive Points
Inverse Distance Weighted
3D Scatter Point Commands

The following tools are active in the dynamic portion of the Tool Palette whenever the 3D Scatter Point Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window with the cursor depends on the current tool. The table below describes the tools in the 3D Scatter Set tool palette.

Tool	Tool Name	Description
	Select Scatter Point	The Select Scatter Point tool is used to select individual scatter points for editing using the Edit Window. Scatter points can also be dragged with the mouse. Scatter points can be deleted. With extremely large sets of scatter points, it may become difficult to identify a scatter point with a particular ID, even if the scatter point IDs are being displayed. In such cases, the ***Find Point*** command in the *Scatter Points* menu can be used to quickly locate a point. The command prompts the user for the ID of the desired point and the point is selected.
	Select Scatter Point Set	The Select Scatter Point Set tool is used to select entire scatter point sets for deletion or to designate the active scatter point set. When this tool is active, an icon appears at the centroid of the set for each of the scatter point sets. A scatter point set is selected by selecting the icon for the set.
	Create Scatter Point	This tool is used to interactively create scatter points by clicking in the GMS graphics window.

Interpolating with 3D Scatter Points

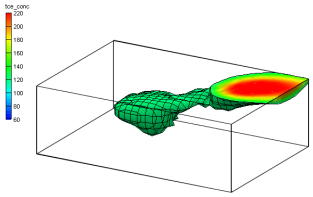
3D Scatter Point Module

3D Scatter Point
Interpolating with 3D Scatter Points
Converting 3D Scatter Points to Other Data Types
Bounding Grid
More
3D Scatter Point Display Options
3D Scatter Point Tool Palette
Active/Inactive Points
Inverse Distance Weighted
3D Scatter Point Commands

Scatter point sets are used for interpolation to other data types such as TINs, grids, and meshes. A 3D grid can be created which will just enclose the scatter points by using the ***Bounding Grid*** command in the *Scatter Points* menu. Interpolation is useful for such tasks as contouring or setting up input data to a model. Since no interpolation scheme is superior in all cases, several interpolation techniques are provided in GMS.

The basic approach to performing an interpolation is to select an appropriate interpolation scheme and interpolation parameters, and then interpolate to the desired object using one of the 3D Interpolation Commands.

The interpolation options are selected using the Interpolation Options dialog accessed through the ***Interp. Options*** command in the *Interpolation* menu. Once a set of options is selected, those options are used for all subsequent interpolation commands. Interpolation options are saved in the project file.

Converting 3D Scatter Points to Other Data Types

3D Scatter Point Module

3D Scatter Point
Interpolating with 3D Scatter Points
Converting 3D Scatter Points to Other Data Types
Bounding Grid
More
3D Scatter Point Display Options
3D Scatter Point Tool Palette
Active/Inactive Points
Inverse Distance Weighted
3D Scatter Point Commands

3D Scatter Points may be converted to 3D Mesh Nodes or Observation Points by using the following commands available either in the *Scatter Points* menu or by right clicking on a scatter set in the Project Explorer:

Scatter Points → Mesh Nodes

The ***Scatter Points → Mesh Nodes*** command is used to convert each of the scatter points to a 3D mesh node. The nodes can then be used to generate a mesh using the ***Tessellate*** command in the *Mesh* menu. The *Mesh* menu can be made visible by selecting the 3D Mesh module.

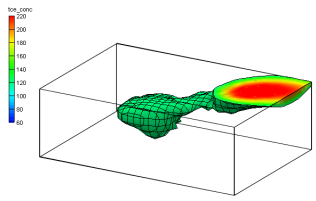
Scatter Points → Obs. Pts.

The ***Scatter Points → Obs. Pts.*** command in the *Scatter Points* menu of the 3D Scatter Point module creates a new observation coverage with one observation point for each of the 3D scatter points in the active scatter point set. The measured values for the new observation points are taken from the scatter set's active dataset values.

MODPATH Starting Locations

The ***Convert To → MODPATH Starting Locations*** command creates MODPATH starting locations from the 3D scatter sets selected in the Project Explorer. A new particle set is created for each 3D scatter point set. This command is available by right-clicking on a 3D scatter set in the Project Explorer. MODPATH starting locations can also be converted to 3D scatter points via the MODPATH menu.

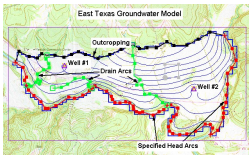
Bounding Grid

3D Scatter Point Module

3D Scatter Point
Interpolating with 3D Scatter Points
Converting 3D Scatter Points to Other Data Types
Bounding Grid
More
3D Scatter Point Display Options
3D Scatter Point Tool Palette
Active/Inactive Points
Inverse Distance Weighted
3D Scatter Point Commands

In many cases, it is useful to interpolate to a 3D grid which just contains the points from a 3D scatter set. The ***Bounding Grid*** command was designed in order to simplify the creation of such a grid. Selecting the ***Bounding Grid*** command from the *Scatter Points* menu brings up the *Create Grid* dialog pre-filled with grid dimensions set at 10% beyond the bounds of the active scatter point set.

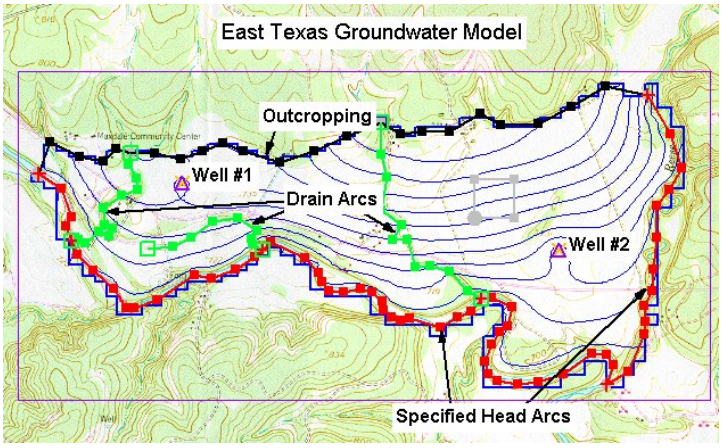
5.10. Map Module

Map Module

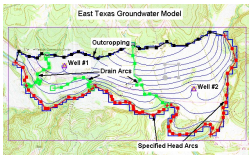
Map Module

Map
Feature Objects
Conceptual Model
Coverages
Clean Command
More
Feature Object Display Options
Feature Object Tool Palette
GIS Module
Images
Map Module Commands

The Map module provides a suite of tools for using Feature Objects to build conceptual models.

Feature objects are used to provide some GIS-like capabilities within GMS. Feature objects include points, arcs, and polygons. Feature objects can be grouped into layers or coverages. A set of coverages can be constructed representing a conceptual model of a groundwater modeling problem. This high level representation can be used to automatically generate MODFLOW and MT3DMS numerical models. Feature objects can also be used for automated mesh generation.



Feature Objects

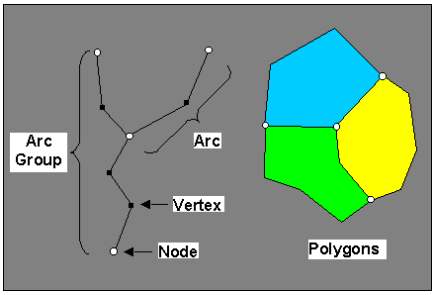
Map Module

Map
Feature Objects
Conceptual Model
Coverages
Clean Command
More
Feature Object Display Options
Feature Object Tool Palette
GIS Module
Images
Map Module Commands

Feature objects in GMS have been patterned after Geographic Information Systems (GIS) objects and include points, nodes, arcs, and polygons. Feature objects can be grouped together into coverages, each coverage defining a particular set of information. Since feature objects are patterned after GIS objects, it is possible to import and export feature objects to a GIS such as Arc/Info or ArcView.

The primary use of feature objects is to generate a high level conceptual model representation of a site. In such a model, items such as rivers, drains, wells, lakes are represented with points, arcs, and polygons. Attributes such as conductance, pumping rates, and elevations are defined with the objects. This conceptual model is then used to automatically generate a grid or mesh and assign the boundary conditions and model parameters to the appropriate cells. Thus, the user can focus on a simplified, high level representation of the model and little or no tedious cell-by-cell editing is required. The feature object approach can be used to build models for SEEP2D, FEMWATER, MODFLOW, MT3DMS, RT3D, and SEAM3D. Feature objects are also used to construct cross sections.

Object Types

The definition of feature objects in GMS follows the paradigm used by typical GIS software that supports vector data. The basic object types are points, nodes, vertices, arcs, arc groups, and polygons. The relationship between these objects is illustrated in the following figure.



Feature Object Types.

Points

Points are XY locations that are not attached to an arc. Points have unique IDs and can be assigned attributes. Points are often used to represent wells. Points are also used when importing a set of XY locations for the purpose of creating arcs or polygons.

Arcs

Arcs are sequences of line segments or edges which are grouped together as a single "polyline" entity. Arcs have unique IDs and can be assigned attributes. Arcs are grouped together to form polygons or are used independently to represent linear features such as rivers. The two end points of an arc are called "nodes" and the intermediate points are called "vertices".

Create Arc Group

This command is used to create an arc group from a set of selected arcs. Once the arc group is created, it can be selected using the ***Select Arc Group*** tool. Properties can be assigned to the arc group as a whole, and the arc group can be selected to display the computed flow through the arc group. An arc group is deleted by selecting the arc group and selecting the **Delete** key. Deleting an arc group does not delete the underlying arcs.

Reverse Arc Direction

Each arc has a direction. One node is the "from" node, the other node is the "to" node. For most applications, the direction of the arc does not matter. However, when the arc is used to define a MODFLOW stream network, the direction of the arc becomes significant. The ***Reverse Arc Direction*** command can be used to change the direction (upstream to downstream) for a stream type arc.

Nodes

Nodes define the beginning and ending XY locations of an arc. Nodes have unique IDs and can be assigned attributes.

Vertices

Vertices are XY locations along arcs in between the beginning and ending nodes. They are used solely to define the geometry of the arcs. Vertices do not have IDs or attributes.

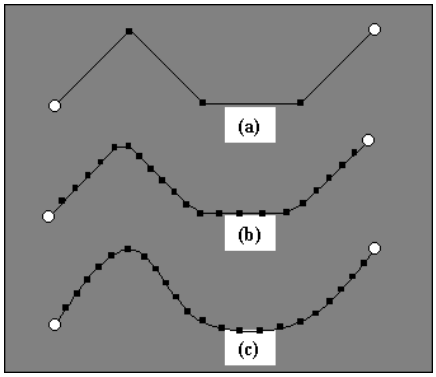
Redistribute Vertices

The primary function of the vertices of an arc is to define the geometry of the arc. In most cases, the spacing of the vertices does not matter. However, if the arcs are to be used for automatic mesh generation, the spacing of the vertices is important. In this case, the spacing of the vertices defines the density of the elements in the resulting mesh. Each edge defined by a pair of vertices becomes the edge of an element. The mesh gradation is controlled by defining closely spaced vertices in regions where the mesh is to be dense and widely spaced vertices in regions where the mesh is to be coarse.

When spacing vertices along arcs, the ***Redistribute vertices*** command in the *Feature Objects* menu can be used to automatically create a new set of vertices along a selected set of arcs at either a higher or lower density. The desired arc should be selected prior to selecting the ***Redistribute vertices*** command. The ***Redistribute vertices*** command brings up the Redistribute Vertices dialog. The following options are available for redistributing vertices:

Linear Interpolation – If the Linear interpolation option is specified, then either a number of subdivisions or a target spacing can be given to determine how points are redistributed along the selected arcs. In either case, the new vertices are positioned along a linear interpolation of the original arc.

Spline Interpolation – If the Spline interpolation option is specified, vertices are redistributed along a series of cubic splines defined by the original vertices of the selected arcs. The difference between the linear and spline interpolation methods is illustrated below:



Redistributing Vertices. (a) Original Arc (b) Linear Interpolation (c) Spline Interpolation.

Vertex to Node

In some cases, it is necessary to split an arc into two arcs. This can be accomplished using the ***Vertex ↔ Node*** command. Before selecting this command, a vertex on the arc at the location where the arc is to be split should be selected. The selected vertex is converted to a node and the arc is split in two.

The ***Vertex ↔ Node*** command can also be used to combine two adjacent arcs into a single arc. This is accomplished by converting the node joining the two arcs into a vertex. Two arcs can only be merged if no other arcs are connected to the node separating the arcs. Otherwise, the node must be preserved to define the junction between the branching arcs.

Arc Groups

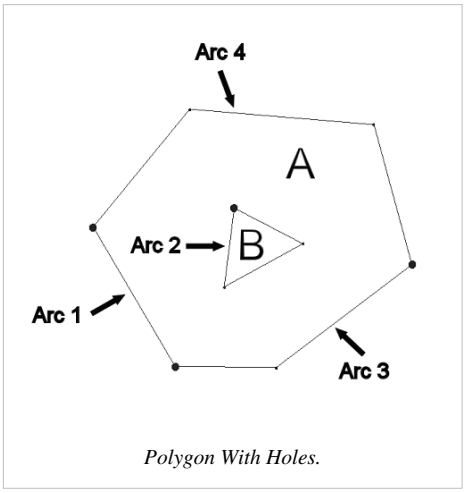
An arc group is a set of arcs that has been marked as a group by the user. As an arc group, attributes can be assigned to the entire group rather than to individual arcs. An arc group can also be selected as a single unit. Arc groups are primarily used for flow observations.

Polygons

Polygons are a group of connected arcs that form a closed loop. A polygon can consist of a single arc or multiple arcs. If two polygons are adjacent, the arc(s) forming the boundary between the polygons is shared (not duplicated).

Polygons may not overlap. However, a polygon can have a hole defined by having a set of closed arcs defining interior polygons. An example of such a case is shown in the figure below where three arcs are used to define two polygons. Polygon A is made up of arcs 1, 2, 3 and 4, whereas polygon B is defined by a single arc (arc 2). For polygon A arcs 1, 3, and 4 define the exterior boundary whereas arc 2 defines a hole.

Polygons have unique IDs and can be assigned attributes. Polygons are used to represent material zones, lakes, variable head zones, etc.



Polygon With Holes.

Build Polygons

While most feature objects can be constructed with tools in the Tool Palette, polygons are constructed with the ***Build Polygons*** command. Since polygons are defined by arcs, the first step in constructing a polygon is to create the arcs forming the boundary of the polygon. Once the arcs are created, they should be selected with the Select Arc tool, and the ***Build Polygons*** command should be selected from the *Feature Objects* menu. If the selected arcs do not form a valid loop, an error message is given.

The ***Build Polygons*** command can be used to construct one polygon at a time or to construct several polygons at once. If the selected arcs form a single loop, only one polygon is created. If the arcs form multiple loops, a polygon is created for each unique (non-overlapping) loop. If no arcs are selected, all of the currently defined arcs in the active coverage are used to create polygons.

Coverages

Feature objects are grouped together into coverages. Each coverage represents a particular set of data. For example, one coverage can be used to define recharge zones, and another coverage can be used to define zones of hydraulic conductivity.

Conceptual Models

In a generic sense, a conceptual model is a simplified, high level model of a site. In GMS, a conceptual model object consists of a set of coverages which are tied to a particular numerical model like MODFLOW or FEMWATER. The coverages below a conceptual model can have attributes that are related to the numerical model. For example, a coverage below a MODFLOW conceptual model can have drain or river arcs.

Feature Object Properties

The Feature Object Properties dialog is used to edit the properties of Points, Nodes, Arcs, Arc Groups, and Polygons. Three filters are located at the top of the dialog.

The Feature type combo box is used to choose which feature (Point, Arcs...) the spreadsheet displays. The Show combo box will show only the selected features or all features depending on which option is selected. The BC type combo box is used to display only certain boundary conditions. For example, if the filter is changed to "well," then only the wells would be displayed in the spreadsheet.

The Show point coordinates toggle is used to display the (x, y, z) coordinates of each point in the spreadsheet. The Add Point and Delete Point buttons are used to create new points or remove points from the coverage.

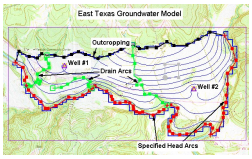
The spreadsheet displays an attribute table associated with the current feature type (Point, Arc...). The columns available in the spreadsheet depend on the options selected in Coverage Setup dialog.

Converting Feature Objects

Feature objects can be converted to other data types in GMS such as cross sections and scatter points. This can be accomplished by either right-clicking on a conceptual model, coverage, grid frame, or by selecting a command from the *Feature Objects* menu. These commands are summarized on the following pages.

- GMS:Map to Modules
- GMS:Map to Models

Conceptual Model

Map Module

Map
Feature Objects
Conceptual Model
Coverages
Clean Command
More
Feature Object Display Options
Feature Object Tool Palette
GIS Module
Images
Map Module Commands

A conceptual model is a group of coverages that are linked to a particular numerical model such as MODFLOW. Once a conceptual model has been defined, coverages can be created beneath the conceptual model. The properties available in the coverage setup dialog depend on the model associated with the conceptual model.

Conceptual Model Properties

Each conceptual model has a name and a numerical model. Then depending on the numerical model other properties can be assigned. The following is a list of the numerical model and additional properties that are assigned to the conceptual model.

MODFLOW – The flow package can be LPF, BCF, or HUF for a MODFLOW conceptual model. Optionally, transport can be included with the MODFLOW conceptual model. If transport is turned on then the transport model must be selected (MT3DMS, RT3D, SEAM3D), and species and/or reaction parameters need to be entered.

FEMWATER – A FEMWATER conceptual model has the option of simulating flow and/or transport.

MODAEM – No additional properties are set for a MODAEM conceptual model.

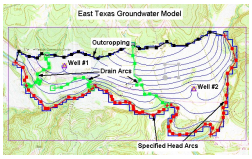
WASH123D – A WASH conceptual model has the option of simulating 3D subsurface flow and/or 3D subsurface tranport and/or 2D overland flow. If the 3D subsurface transport option is turned on, then chemicals must be created.

SEEP2D – No additional properties are set for a SEEP2D conceptual model.

ART3D – Species must be defined with an ART3D conceptual model.

Horizons – No additional properties are set for a Horizon conceptual model.

Feature Object Display Options

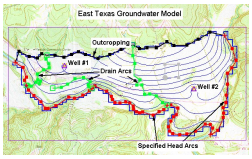
Map Module

Map
Feature Objects
Conceptual Model
Coverages
Clean Command
More
Feature Object Display Options
Feature Object Tool Palette
GIS Module
Images
Map Module Commands

The properties of all feature object and coverage data that GMS displays on the screen can be controlled through the Map tab of the *Display Options* dialog. This dialog is opened by right clicking on the Map Data entry in the Project Explorer and selecting the ***Display Options*** command. It can also be accessed from the from the *Display* menu or the Display Options macro. The objects on the left of the dialog are common to all coverages, regardless of the coverage type, and are always available in the *Display Options* dialog. The options on the right of the dialog depend on the coverage type. The following table describes the general display options available for the Map module.

Display Option	Description
ID	If this option is selected, the ID of each of the feature objects is displayed next to the object. The graphical attributes of the text used to display the IDs are edited using the fields on the right side of the dialog.
Labels	If this option is selected, the name of points is displayed next to the object. The graphical attributes of the text used to display the Labels are edited using the fields on the right side of the dialog.
Points	This option is used to display points. The graphical attributes of the points (symbol, color, size, etc.) depend on the coverage type and are edited using the fields on the right side of the dialog.
Nodes	This option is used to display nodes. The graphical attributes of the nodes (symbol, color, size, etc.) depend on the coverage type and are edited using the fields on the right side of the dialog.
Vertices	This option is used to display the vertices of arcs. A small dot is placed on the arcs at the location of each of the vertices. The color of the vertices is the same as the color of the arcs.
Arcs	This option is used to display arcs. The graphical attributes of the arcs (color, line style, thickness, etc.) depend on the coverage type and are edited using the fields on the right side of the dialog.
Polygons (fill)	If this option is selected, polygons are displayed filled. The graphical attributes of the polygons (fill color) depend on the coverage type and are edited using the fields on the right side of the dialog. The Polygon fill can also be specified as an attribute. Transparency can also be set on the polygon fill using the edit box below the option.

Legend	The Legend item can be used to display a legend listing each of the feature object types being displayed and showing what graphical attributes (symbol, line style, fill color and pattern) are being used to display each type.
Grid frame	This option is used to toggle the display of the Grid Frame.
Show inactive coverages	When several coverages are present, the display of coverages can become confusing. You can choose to not display inactive coverages or change the color attributes on inactive coverages Each of the feature objects in a coverage has a set of display options (color, line style, etc.) that can be edited in the *Display Options* dialog. However, these colors are only used to display the objects in the active coverage. All of the objects in the inactive coverages are displayed using either Coverage colors or the Inactive coverage color depending on the selected option.
Arc direction arrows	This option controls the display of an arrow which shows the arc direction. The pixel length of the arrow can be specified.
Well screens	This options controls the display of the well screens. The width of the well screen can be adjusted in the Width edit field below the toggle.
Calibration targets	This options controls the display of the calibration targets used in the model calibration process. Calibration targets are drawn next to their corresponding map data (point, arc, polygon). The calibration target is drawn such that the height of the target is equal to twice the confidence interval (+ interval on top, – interval on bottom). The Scale edit field allows the user to change the general length and width of the targets independent of the range of the active dataset.
Segment ID	This option controls the display of the segments IDs. The font color and size for the segment can also be adjusted.

Feature Object Tool Palette

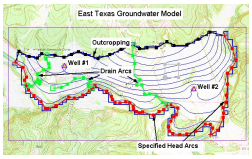
Map Module

Map
Feature Objects
Conceptual Model
Coverages
Clean Command
More
Feature Object Display Options
Feature Object Tool Palette
GIS Module
Images
Map Module Commands

Several tools are provided in the Tool Palette for creating and editing feature objects. These tools are located in the dynamic portion of the Tool Palette and are only available when the Map module is active. The table below describes the feature object tools.

Tool	Tool Name	Description
	Select Tool	Generic selection tool that selects existing feature objects, including: nodes, vertices, arcs, and polygons. A selected object can be deleted, moved to a new location, or operated on by one of the commands in the *Feature Objects* menu. The coordinates of selected points/nodes can be edited using the Edit Window. Double-clicking on a object with this tool brings up that objects attribute dialog.
	Select Point/Node	Selection tool that will only select existing points or nodes. A selected point/node can be deleted, moved to a new location, or operated on by one of the commands in the *Feature Objects* menu. The coordinates of selected points/nodes can be edited using the Edit Window. Double-clicking on a point or node with this tool brings up the Point or *Node Attribute* dialog.
	Select Vertex	Selection tool that will only select existing vertices on arcs. Once selected, a vertex can be deleted, moved to a new location, or operated on by one of the commands in the *Feature Objects* menu. The coordinates of a selected vertex can be edited using the Edit Window.
	Select Arc	Selection tool that will only select existing arcs to perform operations such as deletion, redistribution of vertices, or building polygons. Double-clicking on an arc with this tool brings up the *Arc Attributes* dialog.
	Select Arc Group	Selection tool that is used to select an arc group to assign attributes or to display the computed flux on the arc group. An arc group is created by selecting a set of arcs and selecting the ***Create Arc Group*** command. An arc group is deleted by selecting the arc group and selecting the **Delete** key or by selecting the ***Delete*** command in the *Edit* menu. Deleting an arc group does NOT delete the underlying arc objects.
	Select Polygon	Selection tool that will only select previously created polygons for operations such as deletion, assigning attributes, etc. A polygon is selected by clicking anywhere in the interior of the polygon. Double-clicking on a polygon with this tool brings up the *Polygon Attributes* dialog.
	Select Grid Frame	Selection tool used to select grid frames, allowing for the editing of the grid frame. Once the grid frame is selected, the placement and size of the grid frame can be edited by clicking on small rectangles and dragging.

	Create Point	Creates new points. A new point is created for each location the cursor is clicked on in the Graphics Window. Once the point is created, it can be repositioned or otherwise edited with the Select ***Point/Node*** tool.
	Create Vertex	Creates new vertices along existing arcs. This is typically done to add more detail to the arc. A new vertex is created for each location the cursor is clicked on in the Graphics Window that is within a given pixel tolerance of an existing arc. Once the vertex is created, it can be repositioned with the ***Select Vertex*** tool.
	Create Arc	Creates new arcs. An arc is created by clicking once on the location where the arc is to begin, clicking once to define the location of each of the vertices in the interior of the arc, and double-clicking at the location of the end node of the arc. As arcs are created, it is often necessary for the beginning or ending node of the arc to coincide with an existing node. If you click on an existing node (within a given pixel tolerance) when beginning or ending an arc, that node is used to define the arc node as opposed to creating a new node. If you click on a vertex of another arc while creating an arc, that vertex is converted to a node and the node is used in the new arc. If you click within a given tolerance of an arc edge, a new node is inserted in the arc. If you click on an existing point while creating an arc, the point is converted to a vertex, unless it is the beginning or ending location of an arc, in which case it is converted to a node. While creating an arc, it is common to make a mistake by clicking on the wrong location. In such cases, hitting the Backspace key backs up the arc by one vertex. The ESC key can also be used to abort the entire arc creation process at any time.

Coverages

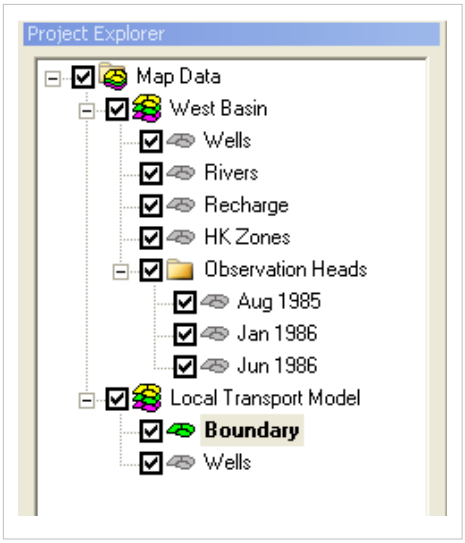
Map Module

Map
Feature Objects
Conceptual Model
Coverages
Clean Command
More
Feature Object Display Options
Feature Object Tool Palette
GIS Module
Images
Map Module Commands

Feature Objects in the Map module are grouped into coverages. Coverages are grouped into conceptual models.

A coverage is similar to a layer in a CAD drawing. Each coverage represents a particular set of information. For example, one coverage could be used to define recharge zones and another coverage could be used to define zones of hydraulic conductivity. These objects could not be included in a single coverage since polygons within a coverage are not allowed to overlap and recharge zones will typically overlap hydraulic conductivity zones.

Coverages are managed using the Project Explorer. Coverages are organized below conceptual models. When GMS is first launched, no coverage exists. If no coverage exists and the user creates feature objects then a new coverage will automatically be created. When multiple coverages are created, one coverage is designated the "active" coverage. New feature objects are always added to the active coverage and only objects in the active coverage can be edited. The figure below shows several coverages in the Project Explorer. The active coverage is displayed with a color icon and bold text. A coverage is made the active coverage by selecting it from the Project Explorer. In some cases it is useful to hide some or all of the coverages. The visibility of a coverage is controlled using the check box next to the coverage in the Project Explorer.

A new coverage can be created by right-clicking on a folder or conceptual model and selecting the ***New Coverage*** command in the pop-up menu.



Right clicking on a coverage brings up a menu with the following options: Delete, Duplicate, Rename, Coverage Setup, Attribute Table, the Map To submenu, Transform, Export, and Properties. The Delete, Duplicate, and Rename commands are self explanatory.

Coverage Setup

The Coverage Setup command brings up the Coverage Setup dialog. This dialog controls the properties that are assigned to feature objects. The feature object properties have been divided into 3 general categories: *Sources/Sinks/BCs*, *Areal Properties*, and *Observation Points*. Under the *Sources/Sinks/BCs* the user can select which source/sinks he would like to include in the coverage (like wells, rivers, drains, etc). *Areal Properties* includes recharge, ET, hydraulic conductivity, and other properties that are assigned to polygonal zones. *Observation Points* control which data sets have associated observation data.

The *Coverage type* is used for WASH123D conceptual models to set the coverage to be a 3D or a 2D coverage.

The *Default layer range* is used with MODFLOW conceptual models to default the "from layer"/"to layer" assignments for boundary conditions.

The *Use to define model boundary* toggle is used with MODFLOW and MODAEM. This means that the polygons in this coverage are used to define the active area of the model.

3D grid layer option for obs. pts

The *3D grid layer option for obs. pts.* is used to set the input option for observations associated with MODFLOW conceptual models. The MODFLOW observation package can handle observations that include multiple cells.Three options are available for determining which layer the observation point will be located:

- by z location – When the "by z location" option is selected, the computed value for the observation point (that will be compared with the observed value) will be taken from the cell that corresponds with the elevation value assigned to the observation point.
- by layer number – If you select the "by layer number" option in the coverage setup, the computed value will be taken from the cell that corresponds to the layer that is specified in the observation point coverage properties.
- Use well screen – This option may be used when the model includes wells with screens. GMS finds the cell or cells that intersect the screened interval the user has entered.

The *Default elevation* field can be used to define the initial Z elevation of new objects created in a coverage. By assigning a different elevation to each of the coverages, the coverages can be displayed as a stack of layers in oblique view.

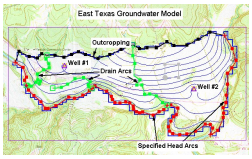
Feature Object Attribute Table

All feature object properties are edited using a single spreadsheet. This makes it possible to cut and paste feature object data using the clipboard and it makes it easier to edit entire columns of data at once. Filters at the top of the dialog control what type of objects are displayed.

Map To Submenu

Coverages can be mapped to other geometric objects or Numerical models by selecting the corresponding command from the Map to Submenu.

Grid Frame

Map Module

Map
Feature Objects
Conceptual Model
Coverages
Clean Command
More
Feature Object Display Options
Feature Object Tool Palette
GIS Module
Images
Map Module Commands

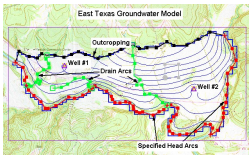
A grid frame is an outline showing where a grid will be created. The grid frame can be used to create a grid at a particular location, size and orientation. The ***Feature Objects | New Grid Frame*** command is used to create the grid frame. When the ***Map → 3D Grid*** command is selected, the grid will be created using the grid frame.

The Grid Frame tool can be used to move, size and rotate the grid frame. Double clicking on the grid frame will bring up the Grid Frame Properties dialog which can also be used to move, size and rotate the grid frame.

Displaying the Grid Frame

The display of the grid frame can be turned on or off by checking (unchecking) the toggle next to the Grid Frame in the Project Explorer or by using the Grid Frame option in the *Feature Objects Display Options* dialog.

Clean Command

Map Module

Map
Feature Objects
Conceptual Model
Coverages
Clean Command
More
Feature Object Display Options
Feature Object Tool Palette
GIS Module
Images
Map Module Commands

The ***Clean*** command is used to fix errors in feature object data. The ***Clean*** command only applies to the active coverage. Selecting the ***Clean*** command brings up the Clean dialog. The clean options are as follows:

- **Snap Nodes –** Any two nodes (or points) separated by a distance which is less than the specified distance tolerance are combined to form a single node.
- **Snap Selected Nodes –** This option is the same as the previous option but only the selected nodes are checked. When this option is checked you will be prompted to select a snapping point; you must click on the graphics window to indicate the snapping point.
- **Intersect Arcs –** All arcs are checked to see if they intersect. If an intersection is found, a node is created at the intersection and the arcs are split into smaller arcs.
- **Intersect Selected Arcs –** This option is the same as the previous option but only selected arcs are checked for intersections.
- **Remove Dangling Arcs –** A check is made for dangling arcs (arcs with one end not connected to another arc) with a length less than the specified minimum length. If any are found they are deleted.

Temporal Discretization

Many of the parameters associated with feature objects can be specified as either constant or transient values. Transient values are defined as a simple list of time/data pairs using the XY Series editor. The time series represents a piece-wise linear curve indicating how the parameter varies with time. When the ***Map → MODFLOW*** command is selected, these curves must undergo temporal discretization. Temporal discretization is a process of converting general time series into discrete values that apply over specific time ranges (stress periods). Transient parameters associated with feature objects are stored in an xy series. An xy series is a general-purpose object used in GMS to represent curves of data (in this case a time series). An xy series is manipulated by GMS with regards to feature objects in three different ways: extrapolation, interpolation, and integration.

Extrapolation

Because the user is free to enter any time values for the x parameter of an xy series, it is possible that the xy series as entered does not cover the same time range as the stress periods. In this case it may be necessary to extrapolate a value for the xy series at a time before or after the first or last entered value. In GMS the simplest approach has been used. If a value is required for a time previous to the times defined by the xy series, the first value is used. Likewise for a time that is later than the all of the times in the xy series, the last value is used. Since this behavior might hide an error in the input parameters, GMS will warn the user if any xy series does not cover the time range defined by the stress periods.

Interpolation

It is also sometimes necessary to create an xy series that is a composite of two other xy series. This is the case when obtaining transient values for an intermediate point along an arc segment that has differing transient parameters at both nodes at the ends of the arc. To perform this type of interpolation, a new xy series is constructed that is the union of the x times from the two original series. The y values that correspond to the time step in each of the series are used to obtain a new y value for the intermediate point:

$$F = (1 - h_1)y_1 + (1 - h_2)y_2$$

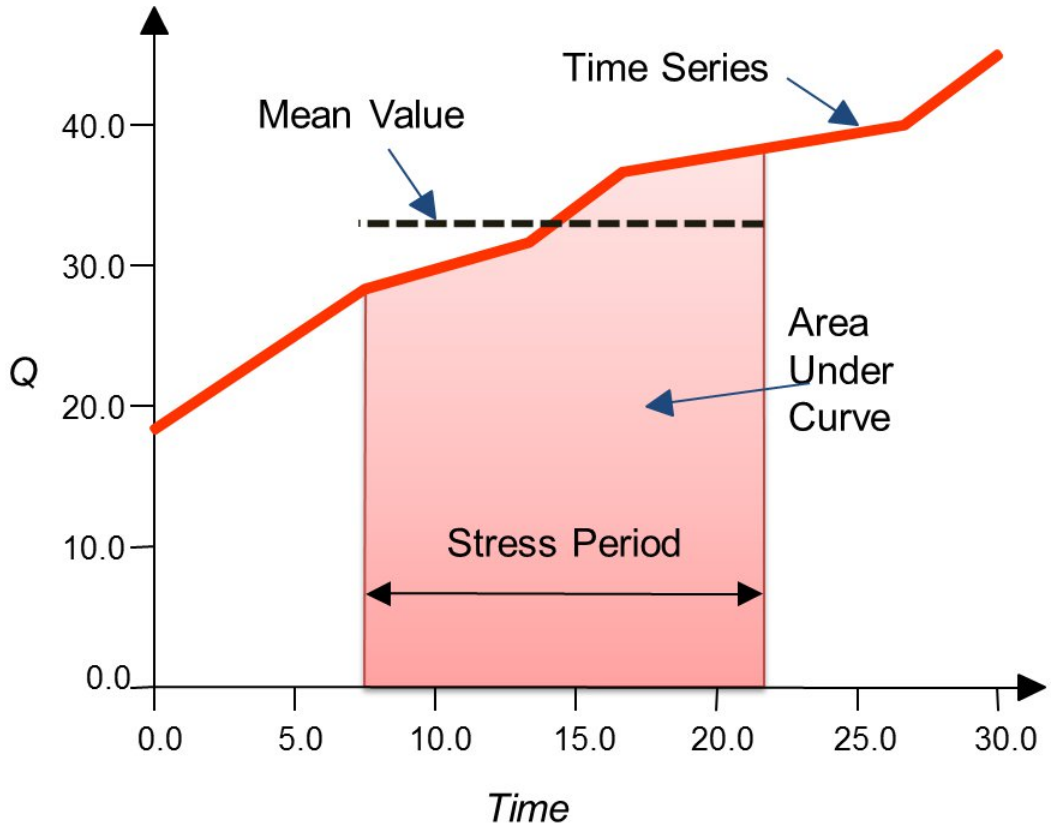
Where F is the y-value along the new xy series, y_1 and y_2 are the two node xy series and h_n is the interpolation weighting parameter.

As an example, when one node of an arc has a constant parameter and the other has a transient parameter, the constant parameter is converted into an xy series with only one point. By using the extrapolation assumption above, it is then possible to perform a transient interpolation using two transient series.

Integration

MODFLOW and MT3DMS both use the concept of Stress Periods to define the times that stresses may be applied. A stress period is a time interval during which all external stresses are constant. Because an xy series is not constrained to be constant over a time interval, it is necessary to obtain a representative value from the xy series that will approximate this condition.

GMS uses integration of the curve defined by the xy series to obtain the average value over the stress period. This average value is then assigned to the stress period.



Time Units

When entering a time series in the Map module using the XY Series Editor, you can use relative times (i.e., 0.0, 3.2, 5.4 etc.), or dates/times (1/1/2004 12:00:00 AM, 2/13/2004 2:01:00 PM etc.).

Map to Models

Map to MODFLOW

See Map to MODFLOW

Map to FEMWATER

Once the 3D mesh is constructed, the final step in converting the FEMWATER conceptual model to a mesh-based numerical model is to select the ***Map → FEMWATER*** command in the *Feature Objects* menu. This command assigns the wells, boundary conditions, and recharge zones assigned to the points, arcs, and polygons in the conceptual model to the nodes and element faces of the 3D mesh. At this point, the basic analysis options (steady state vs. transient, output control, material properties, etc.) must still be assigned using the tools in the *FEMWATER* menu. Once these basic options have been assigned, the model can be saved and FEMWATER can be launched.

Map to MT3DMS

After the conceptual model is constructed, the ***Map → MT3DMS*** command can be used to convert the conceptual model to an MT3DMS numerical model. Before the ***Map → MT3DMS*** command can be selected, the MT3DMS data must be initialized. The MT3DMS data are initialized with the following steps:

1. Switch to the 3D Grid module
2. Select the ***New Simulation*** command from the *MT3D* menu.
3. Open the Basic Transport Package dialog and set up the stress periods you wish to use in the simulation.

Once the MT3DMS data are initialized, the ***Map → MT3DMS*** command becomes undimmed and can be selected.

Because MT3DMS already assumes a default concentration of zero for an unspecified point source sink, GMS does not create a source/sink if the concentration for the feature object has been specified as a constant value of zero.

Map to Modules

Arc → Cross Sections

The default method for generating cross sections through solids, 3D meshes, or 3D grids it to interactively enter a line or a polyline in the Graphics Window while the Make Cross Section tool is active (Solid module, 3D Grid module, 3D Mesh module). This line is then projected perpendicular to the screen (parallel to the viewers viewing angle) and is intersected with the 3D objects to generate the cross section. In some cases, it is useful to precisely locate the cross section. Furthermore, it is often necessary to repeatedly generate a cross section at the same location. In such cases, the ***Arcs → Cross Sections*** command can be used to precisely control the location of a cross section.

When the ***Arcs → Cross Sections*** command is selected, the *Arcs → Cross Sections* dialog appears. The top part of the dialog is used to specify which of the arcs are to be used to create the cross sections. Either all of the arcs are used or only the selected arcs. Since cross sections can be cut through any 3D object, the items in the lower section of the dialog are used to designate which of the 3D objects will be used to cut the cross sections. If one of the types listed does not currently exist, the corresponding item is dimmed.

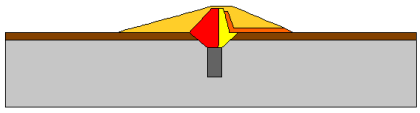
When the OK button is selected, a cross section is constructed for each of the designated arcs. As is the case when the Make Cross Section tool is used, the cross sections are constructed by projecting the arcs parallel to the viewing angle. For example, to create vertical cross sections, the image should be in plan view prior to selecting the ***Arcs → Cross Sections*** command.

Map → TIN

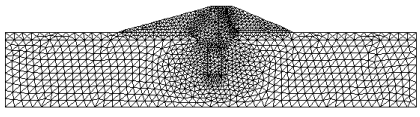
This command creates a TIN using each polygon in the coverage.

Map to 2D Mesh

Once a set of feature objects has been created for a SEEP2D or a FEMWATER conceptual model, the *Map → 2D Mesh* command can be used to generate a 2D finite element mesh from the objects. The *Map → 2D Mesh* command creates a 2D Mesh on the interior of all of the polygons in the current coverage. The figure below shows a cross section of a dam built using the Feature Objects in the Map Module. The second figure shows a 2D Mesh created from the polygons.



Cross Section of a Dam built with Arcs and Polygons in the Map Module



2D Mesh created from Map Objects using Map → 2D Mesh command

Map to 2D Grid

The ***Map → 2D Grid*** command is used to construct a 2D grid using the feature objects in a 2D Grid Coverage. When the ***Map → 2D Grid*** command is selected, the *Create Grid* dialog appears. If a grid frame has been defined, the size and location of the grid frame are used to initialize the fields in the Create Grid dialog. In most cases, these values will not need to be changed and the user can simply select the OK button to create the grid. If a grid frame has not been defined, the size and location of the grid are initialized so that the grid just surrounds the currently defined feature objects. If desired, the grid dimensions can be edited prior to selecting the OK button to create the grid.

If one or more refine points are defined in the conceptual model, the number of rows and columns in the grid will be automatically determined when the grid is created. Thus, these fields cannot be edited by the user and will be dimmed. If refine points are not defined, the user must enter the number of rows and columns.

Map to 3D Grid

Once the feature object coverages defining a conceptual model have been completely defined, the conceptual model is ready to be converted to a numerical model. The first step in this conversion process is to create a grid using the ***Map → 3D Grid*** command. Typically, the ***Grid Frame*** command is used prior to this command to define the location and dimensions of the grid.

When the ***Map → 3D Grid*** command is selected, the *Create Grid* dialog appears. If a grid frame has been defined, the size and location of the grid frame are used to initialize the fields in the *Create Grid* dialog. In most cases, these values will not need to be changed and the user can simply select the OK button to create the grid. If a grid frame has not been defined, the size and location of the grid are initialized so that the grid just surrounds the currently defined conceptual model. Once again, in most cases, no changes will need to be made and the user can typically immediately select the OK button to create the grid.

If one or more refine points are defined in the conceptual model, the number of rows and columns in the grid will be automatically determined when the grid is created. Thus, these fields cannot be edited by the user and will be dimmed. If refine points are not defined, the number of rows and columns must be entered.

Refine Points

Refine attributes are assigned to points or nodes and are used to automatically increase the grid density around a point when the grid is constructed. Although refine attributes may be associated with any point or node, they are usually assigned in conjunction with wells.

Map to 2D Scatter Points

The Map → 2D Scatter Points command creates a scatter point set from the points and nodes and vertices of the current coverage. The process is different for observation coverages and non-observation coverages.

Non-observation Coverages – With non-observation coverages, a single elevation dataset is created for the 2D scatter points representing the Z location of all the points, nodes and vertices.

Observation Coverages – With observation coverages, the *Observation Points → Scatter Points* dialog appears. This dialog allows you to create a dataset for the 2D scatter points from one of the measurements associated with the observation points.

Measurement – A dataset is created for the 2D scatter points from the measurement selected in the dialog. The model associated with the selected measurement (if any) is shown, along with whether the measurement is steady state or transient.

Time Step Times – This section of the dialog is only available if the selected measurement is transient. It allows you to define the number of time steps, and the time step times to be created for the scatter point dataset.

Match all unique times – The Match all unique times option gets the set of unique times from the XY series of all the observation points. This is the union of all the times. If some XY series use dates/times and others don't, this option won't be available. Otherwise, the times in the spreadsheet will be displayed as either dates/times or relative times depending on the XY series. The spreadsheet will not be editable. The Use dates/times toggle will be unavailable but set according to whether the observation point XY series use dates/times or not. The Reference time section will be unavailable, but if the XY series use dates/times, the minimum time will be used as the reference time for the scatter point dataset.

Match time steps from model – The Match time steps from model option will only be available if the measurement is associated with a model, and the model is transient. If so, this will be the default choice and GMS will get the times to display in the spreadsheet from the stress period and time step info for the model. The spreadsheet will not be editable. The Use dates/times toggle will be unavailable but set according to whether the model uses dates/times or not. The Reference time section will be unavailable, but if the model uses dates/times, the model reference time will be used as the reference time.

Specify times – The spreadsheet of times will be editable with this option and you can copy and paste times from another program such as a spreasheet. Also, the Initialize Times button becomes available allowing you to bring up a dialog you can use to create times at a specified interval. If you select the Use dates/times toggle, the Reference time section will become available and the times in the spreadsheets will be displayed as dates/times.

Map to 3D Scatter Points

The Map → 3D Scatter Points command was introduced in GMS version 8.3. This command works exactly like the Map → 2D Scatter Points command except that 3D scatter points are created. One use of the Map → 3D Scatter Points command is to create scatter points from map data that can then be converted into MODPATH starting locations.

5.11. GIS Module

GIS Module

GIS Module
GIS
Enabling ArcObjects
Converting GIS Data to Other Data Types
Add Data
More
GIS Display Options
GIS Tool Palette
Images
Rasters
GIS Commands

The GIS module allows you to manipulate ESRI type GIS data, such as shapefiles. If you have a license of ArcView 8.x or higher on the computer that GMS is installed on, there are additional features available in the GIS module, but you don't have to have an ArcView 8.x license installed to access the basic features.

The following table shows what features are available with and without a license of ArcView 8.x.

Feature	With ArcView	Without ArcView
Efficient management of large data sets	X	X
Graphical selection of features	X	X
Conversion of selected GIS objects to GMS feature objects	X	X
Viewing attribute tables	X	X
Joining additional attribute tables based on a key field	X	X
Display like ArcView	X	X
Display in a simplified, single color	X	X
Support for shapefiles	X	X
Support for coverages, geodatabases, images, CAD, grids	X	
Selection queries	X	
Symbology	X	

GIS Display Options

GIS Module
GIS
Enabling ArcObjects
Converting GIS Data to Other Data Types
Add Data
More
GIS Display Options
GIS Tool Palette
Images
Rasters
GIS Commands

The properties of all GIS data that GMS displays on the screen can be controlled through the Shape Files tab of the Display Options dialog. This dialog is opened by right clicking on the GIS Data entry in the Project Explorer and selecting the Display Options command. It can also be accessed from the from the Display menu or the Display Options macro. The following table describes the display options available for the GIS module.

Display Option	Description
Points	Controls the display of points in shape files in the Graphics Window. The size and color of the points can be adjusted using the button to the right of the Points toggle.
Lines	Controls the display of lines in shape files in the Graphics Window. The line type, width, and color can be adjusted using the button to the right of the Lines toggle.
Polygons	Controls the display of polygons in shape files in the Graphics Window. The polygon line type, width, and color can be adjusted using the button to the right of the Polygons toggle.
Selection color	Controls the selection color used when GIS shape file objects are selected in the Graphics Window.

GIS Tool Palette

GIS Module
GIS
Enabling ArcObjects
Converting GIS Data to Other Data Types
Add Data
More
GIS Display Options
GIS Tool Palette
Images
Rasters
GIS Commands

The following tools are active in the dynamic portion of the Tool Palette whenever the GIS module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window with the cursor depends on the current tool. The table below describes the tools in the GIS tool palette.

Tool	Tool Name	Description
	Select Features	The ***Select Features tool*** is used to select feature objects in the Graphics Window. This tool is only available if ArcView is installed locally on the user's desktop.
	Identify	The Identify tool is used to select GIS objects and display information relating to that object in an Identify Results dialog. This tool is only available if ArcView is installed locally on the user's desktop.
	Select Elements	The ***Select Elements*** tool is used to select shape objects in the Graphics Window.

Enabling ArcObjects

GIS Module
GIS
Enabling ArcObjects
Converting GIS Data to Other Data Types
Add Data
More
GIS Display Options
GIS Tool Palette
Images
Rasters
GIS Commands

ArcObjects is a development platform provided by ESRI that allows developers of other applications (such as GMS) to incorporate ArcView/ArcGIS capability directly within their application. GMS can use ArcObjects to access some of the same functionality in GMS that is available in ArcView, providing you are running on a computer that has a current license of ArcView.

The ***Data | Enable ArcObjects*** command queries the ESRI license manager for ArcView/ArcGIS to see if a license exists. If a valid license is found then the ArcView functionality within GMS is enabled and you will be allowed access. If a license is not found then the ArcView specific features remain unavailable.

ArcObjects® is a development platform provided by ESRI® [1] that allows developers of other applications to incorporate ArcView/ArcGIS® capability directly within their application. ArcObjects® is used to incorporate ArcView® functionalities into XMS software. This allows users to use ArcView® functionality within XMS software. In order to use ArcView® functionality, a current license of ArcView® must be installed. Without a license, much of the same functionality is available, the primary differences being that only the shapefile format is supported, and many of the selection and display capabilities are minimal.

"Error initializing EMRL_LicCheckMod.dll"

If the dll "EMRL_LicCheckMod.dll" fails to register automatically, selecting the Enable ArcObjects command will bring up the error "Error initializing EMRL_LicCheckMod.dll". Please contact Technical Support". To fix this, you will need to register the dll manually by following the steps below.

- 1. Select "Start | Run"
- 2. Type in 'regsvr32 "<directory where GMS 7.0 was installed>\EMRL_LicCheckMod.dll"

```
The default location for GMS 7.0 is "C:\Program Files\GMS 7.0".  If the program was installed in the default location, for example, this line in the Run window will be: regsvr32 "C:\Program Files\GMS 7.0\EMRL_LicCheckMod.dll"
```

- 3. Select OK
- 4. Restart GMS 7.0

References

[1] http://www.esri.com/

GIS to Feature Objects

GIS Module
GIS
Enabling ArcObjects
Converting GIS Data to Other Data Types
Add Data
More
GIS Display Options
GIS Tool Palette
Images
Rasters
GIS Commands

GIS vector data can be converted to Feature Objects (points, arcs, polygons in coverages) for use in a conceptual model. Attributes associated with the GIS data can also be mapped over to attributes associated with Feature Objects. Two commands are available for converting GIS vector data to Feature Objects: ***GIS | ArcObjects → Feature Objects***(with an ArcObjects license) and ***GIS | Shapes → Feature Objects***. Selecting either of these commands will bring up GIS to Feature Objects wizard.

Before starting the wizard, you need to make sure to set the appropriate coverage as the active coverage since the wizard will convert the GIS data to new Feature Objects in the active coverage. Also, if you are mapping over attributes, you must make sure the GMS coverage attributes are defined before you do the conversion.

If you only wish to convert a portion of you GIS vector data to Feature Objects then select the desired GIS data before beginning the wizard. If you wish to map all the features you can begin the wizard and you will be asked if you want to convert all features since none are selected.

Follow the instructions in the mapping wizard to convert the GIS data to Feature Objects.

Add Data

GIS Module
GIS
Enabling ArcObjects
Converting GIS Data to Other Data Types
Add Data
More
GIS Display Options
GIS Tool Palette
Images
Rasters
GIS Commands

Add Shapefile Data

The ***Data | Add Shapefile Data*** command allows you to browse for and open shapefiles as GIS layers in GMS. Without a license of ArcView on your computer then shapefiles are the only supported format for GIS layers. With a valid license of ArcView the ***Data | Add Data*** command is available and any of the ESRI supported formats can be opened as GIS data layers.

Add Data

The ***Data | Add Data*** command is available when ArcObjects is enabled and uses the same dialog resource to open GIS data layers that is used by ArcView.

When ArcObjects is enabled you are able to load any of the ESRI supported formats, including shapefiles, coverages, geodatabases, grids, images, CAD files and others, as GIS data layers in GMS. These data can then be converted to GMS feature objects in map coverages.

Arc Hydro Groundwater

Arc Hydro Groundwater [1] is a data model and a suite of tools for groundwater data that is integrated with ESRI's [1] ArcGIS [2]. GMS can import and export data in a geodatabase consistent with the Arc Hydro Groundwater standard.

Export to Arc Hydro Groundwater

To export data from GMS into an Arc Hydro Groundwater geodatabase you select the ***File | Save As*** menu command and change the file type to Arc Hydro Groundwater Geodatabase. In order for the export to be successful the following things must be true:

- ArcObjects must be enabled.
- Your data must be in a non-local projection.

Import from Arc Hydro Groundwater

An Arc Hydro Groundwater geodatabase may be imported into GMS. This is done by enabling ArcObjects and, in the GIS module, selecting the ***GIS | Add Data*** menu command. Selecting the Arc Hydro Groundwater geodatabase from the Add Data dialog will display the items the geodatabase contains. You may then select which items to import into GMS.

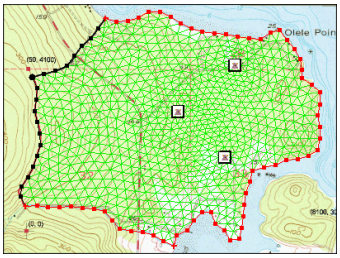
References

[1] http://www.archydrogw.com/ahgw/Main_Page
[2] http://www.esri.com/software/arcgis/

6. Models

6.1. FEMWATER

FEMWATER

FEMWATER	
 *FEMWATER Screenshot*	
Model Info	
Model type	3D mesh, finite-element, Steady state and transient
Developer	George Yeh
Documentation	FEMWATER Manual [1]
Tutorials	FEMWATER Tutorials

GMS includes a graphical interface to the groundwater model FEMWATER. FEMWATER is a 3D finite element, saturated/unsaturated, density driven, flow and transport model. FEMWATER was originally written by G.T. (George) Yeh at Penn State University (Yeh, et. al., 1992). The version of FEMWATER that is supported by GMS is a special version that has been modified by G.T. Yeh and the U.S. Army Engineer Waterways Experiment Station. This version is a coupled version of the original FEMWATER model (which solved for flow only) with the transport model LEWASTE (also developed by George Yeh).

A separate reference manual is available which describes the new version of FEMWATER in detail (Lin, et. al., 1997). This manual contains a description of the input requirements and should be read completely before using GMS to set up a problem.

Pre-Processing

- Building a FEMWATER Model
- FEMWATER Model Input
- Saving a FEMWATER Simulation

Post-Processing

- FEMWATER Display Options
- FEMWATER Post Processing Viewing Options

External Links

- Mar 2004 ERDC/CHL CHETN-XI-1 DDJC-Sharpe Defense Distribution Depot: FEMWATER 3D Transport Model of TCE Plume Migration with Natural Attenuation [2]

References

[1] http://gmsdocs.aquaveo.com/Femwater.pdf
[2] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-xi-1.pdf

6.1.1. FEMWATER Pre-Processing

Building a FEMWATER Model

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Three various approaches provided in GMS for constructing a FEMWATER model. The model can be completely defined using the tools in the 3D Mesh module (the direct approach), or the model can be defined with the aid of the feature object tools in the Map module (the conceptual model approach). A horizons to 3D mesh approach can also be used.

Basic Steps in Building a FEMWATER Model

The basic steps in building a FEMWATER model include:

- Build a 3D mesh defining the extents of the model. Three approaches are available:
  - FEMWATER Direct Approach
  - FEMWATER Conceptual Model Approach
  - Horizons to 3D Mesh
- Initialize FEMWATER
- Assign boundary conditions and model stresses, including sources and sinks
  - Point Source/Sink BC (Wells)
  - Assign Node/Face BC
- Assign material properties
- Run FEMWATER

FEMWATER Direct Approach

The Direct Approach

For models with simple geometry and boundary conditions, the entire model can be constructed using the tools and commands in the 3D Mesh module. With this approach, the editing of the FEMWATER data is performed directly on the nodes and elements of the mesh. The first step is to create a 3D mesh covering the model domain using the mesh building tools in the 3D Mesh module. The boundary conditions and source/sink terms are then assigned by

selecting nodes, elements, and element faces and assigning values directly to the selected objects. The model is then saved and FEMWATER is launched.

Creating a Mesh

The first step in performing a FEMWATER simulation is to create a 3D finite element mesh. The volumetric domain to be modeled by FEMWATER is idealized and discretized into hexahedra, prisms, tetrahedra, and or pyramids. Elements are grouped into zones representing hydrostratigraphic units. Each element is assigned a material ID representing the zone to which the element belongs. When constructing a mesh, care should be taken to ensure that elements do not cross or straddle hydrostratigraphic boundaries.

The tools provided in GMS for constructing a 3D finite element mesh are provided in the 3D Mesh Module. When constructing a mesh for FEMWATER, there are a few important guidelines that should be considered. These guidelines are described in Chapter 3 of the FEMWATER Reference Manual.

The most efficient method for constructing a 3D mesh for FEMWATER is to use the conceptual model approach. The FEMWATER conceptual model can be used to automatically build a 2D mesh that matches the model boundaries and is refined around wells. This mesh can then be extruded into a 3D mesh.

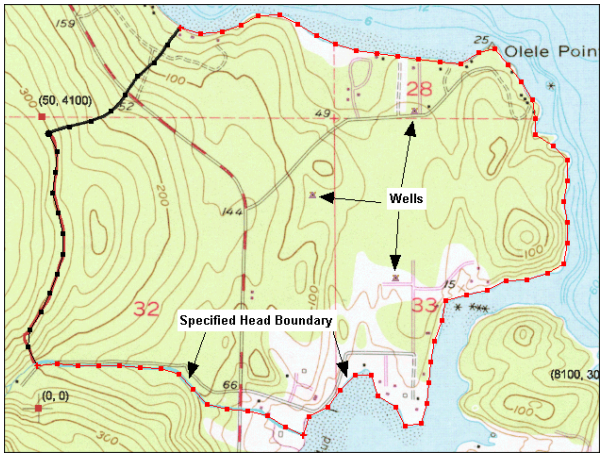
Solids can also be used to make a structured mesh. This is accomplished with the Solids -> Layered Mesh command in the ***Solids*** menu.

FEMWATER Conceptual Model Approach

The preferred method for setting up a FEMWATER simulation is to use the feature object tools in the Map module to define a FEMWATER conceptual model of a site being studied. The conceptual model is a high-level description of the site including sources/sinks, the boundary of the domain to be modeled, rainfall and seepage zones, and material zones within each of the layers. The conceptual model is defined with feature objects, including points, arcs, and polygons, and is constructed independently of a numerical grid. Once the conceptual model is complete, a mesh is automatically constructed to fit the conceptual model, and the FEMWATER data are converted from the conceptual model to the nodes, elements, and element faces. The dialogs and interactive editing tools in the FEMWATER menu can then be used to edit or review the data if desired.

A FEMWATER model can be created in GMS using one of two methods: assigning and editing values directly to the nodes and elements of a mesh (the direct approach), or by constructing a grid-independent representation of the model using feature objects and allowing GMS to automatically assign the values to the nodes and elements (the conceptual model approach). Except for simple problems, the conceptual model approach is typically the most effective.

The conceptual model approach utilizes feature objects in the Map module. A FEMWATER conceptual model is created in the Map module and feature objects are used to create a high level representation of the site you are modeling. The figure below shows a sample conceptual model.



Sample FEMWATER Conceptual Model

Two Step Process

A FEMWATER conceptual model is used to build a numerical model using a two step process. In the first step, a 3D mesh is created. This can be done by using the feature objects in conjunction with a set of TINs to build a 3D Mesh or a solid can be converted to a layered 3D Mesh using the ***Solids -> Layered Mesh*** command. In the second step, the boundary conditions and recharge values assigned to the feature objects are automatically assigned to the appropriate nodes and element faces of the 3D mesh using the ***Feature Objects | Map -> FEMWATER*** command.

To create a FEMWATER conceptual model right click on the Map Data folder in the Project Explorer and select the ***New Conceptual Model*** command. In the Conceptual Model Properties dialog change the model type to FEMWATER in the pull-down list. In the dialog the simulation options of flow and transport can also be toggled on or off. Next create a coverage by right clicking on the FEMWATER conceptual model in the data tree and selecting ***New Coverage***. The coverage attributes can then be setup in the *Coverage Setup* dialog.

A FEMWATER coverage can contain points, arcs, and polygons. The points are used to define wells, the arcs are used to define boundary conditions, and the polygons are used to define recharge zones. In most cases, a single coverage is sufficient. However, multiple FEMWATER coverages can be used if desired.

More on point/arc properties here: FEMWATER Point / Arc Properties

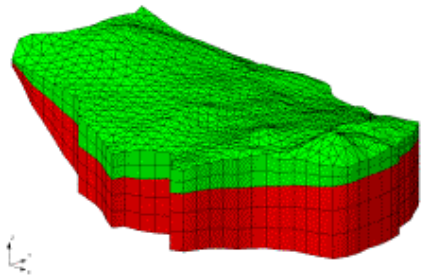
Build a 3D Finite Element Mesh

Once the FEMWATER conceptual model is constructed, the next step is to use the conceptual model to build a 3D finite element mesh. This is accomplished by first building a 2D mesh, then building the 3D mesh by extruding each of the 2D elements in 3D elements.

Map -> 2D Mesh

The first step in building the 3D mesh is to select the *Feature Objects | Map -> 2D* Mesh command. This command creates a 2D mesh by automatically filling in the interior of the conceptual model with nodes and elements. The size and spacing of the elements is controlled by the spacing of the vertices on the arcs and by the refine point attribute assigned to any wells in the interior of the conceptual model.

An example of the ***Map -> 2D Mesh*** command is shown in the following figure. A sample FEMWATER conceptual model is shown in part a. The 2D mesh resulting from execution of the ***Map -> 2D Mesh*** command is shown in part b.



Converting a FEMWATER Conceptual Model (a) Conceptual Model (b) 2D Mesh Created with *Map -> 2D Mesh Command* (c) 3D Mesh Created by Extruding 2D Mesh (d) 3D Mesh after *Map -> FEMWATER* Command.

Creating the 3D Elements

Once the 2D mesh is created, the next step is to create the 3D mesh by extruding each of the 2D elements into a series of 3D elements. The elevations of the 3D elements can be defined from a set of boreholes, a set of TINs, or a set of Solids.

For sites with relatively simple stratigraphy, the ***Regions -> 3D Mesh*** command in the Borehole module can be used.

For sites with more complex stratigraphy, the ***TINs -> 3D Mesh*** command in the TIN module should be used.

A mesh can also be created using the ***Solids -> Layered Mesh*** command in the Solid module.

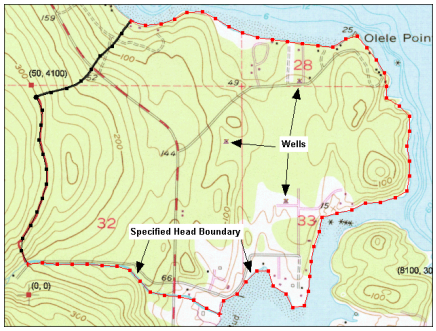
FEMWATER Conceptual Model Approach

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

The preferred method for setting up a FEMWATER simulation is to use the feature object tools in the Map module to define a FEMWATER conceptual model of a site being studied. The conceptual model is a high-level description of the site including sources/sinks, the boundary of the domain to be modeled, rainfall and seepage zones, and material zones within each of the layers. The conceptual model is defined with feature objects, including points, arcs, and polygons, and is constructed independently of a numerical grid. Once the conceptual model is complete, a mesh is automatically constructed to fit the conceptual model, and the FEMWATER data are converted from the conceptual model to the nodes, elements, and element faces. The dialogs and interactive editing tools in the *FEMWATER* menu can then be used to edit or review the data if desired.

A FEMWATER model can be created in GMS using one of two methods: assigning and editing values directly to the nodes and elements of a mesh (the direct approach), or by constructing a grid-independent representation of the model using feature objects and allowing GMS to automatically assign the values to the nodes and elements (the conceptual model approach). Except for simple problems, the conceptual model approach is typically the most effective.

The conceptual model approach utilizes feature objects in the Map module. A FEMWATER conceptual model is created in the Map module and feature objects are used to create a high level representation of the site you are modeling. The figure below shows a sample conceptual model.



Sample FEMWATER Conceptual Model

Two Step Process

A FEMWATER conceptual model is used to build a numerical model using a two step process. In the first step, a 3D mesh is created. This can be done by using the feature objects in conjunction with a set of TINs to build a 3D Mesh or a solid can be converted to a layered 3D Mesh using the ***Solids -> Layered Mesh*** command. In the second step, the boundary conditions and recharge values assigned to the feature objects are automatically assigned to the appropriate nodes and element faces of the 3D mesh using the ***Feature Objects | Map -> FEMWATER*** command.

To create a FEMWATER conceptual model right click on the Map Data folder in the Project Explorer and select the ***New Conceptual Model*** command. In the Conceptual Model Properties dialog change the model type to FEMWATER in the pull-down list. In the dialog the simulation options of flow and transport can also be toggled on or off. Next create a coverage by right clicking on the FEMWATER conceptual model in the data tree and selecting **New Coverage**. The coverage attributes can then be setup in the *Coverage Setup* dialog.

A FEMWATER coverage can contain points, arcs, and polygons. The points are used to define wells, the arcs are used to define boundary conditions, and the polygons are used to define recharge zones. In most cases, a single coverage is sufficient. However, multiple FEMWATER coverages can be used if desired.

More on point/arc properties here: FEMWATER Point / Arc Properties

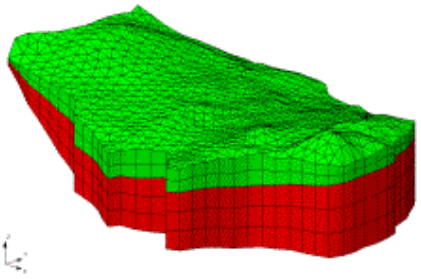
Build a 3D Finite Element Mesh

Once the FEMWATER conceptual model is constructed, the next step is to use the conceptual model to build a 3D finite element mesh. This is accomplished by first building a 2D mesh, then building the 3D mesh by extruding each of the 2D elements in 3D elements.

Map -> 2D Mesh

The first step in building the 3D mesh is to select the ***Feature Objects | Map -> 2D Mesh*** command. This command creates a 2D mesh by automatically filling in the interior of the conceptual model with nodes and elements. The size and spacing of the elements is controlled by the spacing of the vertices on the arcs and by the refine point attribute assigned to any wells in the interior of the conceptual model.

An example of the ***Map -> 2D Mesh*** command is shown in the following figure. A sample FEMWATER conceptual model is shown in part a. The 2D mesh resulting from execution of the ***Map -> 2D Mesh*** command is shown in part b.



Converting a FEMWATER Conceptual Model (a) Conceptual Model (b) 2D Mesh Created with ***Map -> 2D Mesh*** Command (c) 3D Mesh Created by Extruding 2D Mesh (d) 3D Mesh after Map -> FEMWATER Command.

Creating the 3D Elements

Once the 2D mesh is created, the next step is to create the 3D mesh by extruding each of the 2D elements into a series of 3D elements. The elevations of the 3D elements can be defined from a set of boreholes, a set of TINs, or a set of Solids.

For sites with relatively simple stratigraphy, the ***Regions -> 3D Mesh*** command in the Borehole module can be used.

For sites with more complex stratigraphy, the ***TINs -> 3D Mesh*** command in the TIN module should be used.

A mesh can also be created using the ***Solids -> Layered Mesh*** command in the Solid module.

FEMWATER Flows

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Computed flows can be automatically summed and displayed for FEMWATER simulations. To enable this option, the Save flux file option must be selected in the FEMWATER Output Control dialog prior to saving the FEMWATER model. When this option is selected, FEMWATER saves a lumped nodal flow data set file as part of the FEMWATER solution. This file is automatically read into GMS as part of the FEMWATER solution.

Once a FEMWATER flux file has been read into memory as part of a FEMWATER solution, the computed flux through a set of nodes can be displayed simply by selecting the nodes. The flow budget (in, out, net) for the selected nodes is displayed in the ***Status Bar*** at the bottom of the GMS window. The flow budget is only displayed if the active solution is a FEMWATER solution.

It should be noted that lumped nodal fluxes are only non-zero for boundary nodes where a boundary condition has been assigned.

Point Source/Sink BC

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

The ***Point Source/Sink BC*** command is used to assign a flow rate to a node. This option is typically used to assign flow rates to interior nodes to simulate injection or extraction wells. When a point source/sink is first assigned to a node, the node should be selected with the ***Select Nodes*** tool. The ***Point Source/Sink BC*** command is then selected and the Point Source/Sink BC dialog appears. Both a flow rate and a concentration may be specified at a point source/sink node. The values can be constant or transient.

Existing boundary conditions can be deleted by selecting the boundary condition with the ***Select Boundary Nodes***, ***Select Boundary Faces***, or ***Select Wells*** tool and selecting the ***Delete BC*** command from the *FEMWATER* menu.

The Select Well Tool

When a point source/sink boundary condition is applied to a node, a well symbol is placed on the node. The ***Select Wells*** tool can then be used to select the node rather than the ***Select Nodes*** tool whenever the well needs to be edited or deleted. The ***Select Wells*** tool is easier to use than the ***Select Nodes*** tool when there are a large number of nodes since it only selects nodes with point source/sink boundary conditions.

Assign Node/Face BC

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

The ***Assign Node/Face BC*** command in the *FEMWATER* menu is used to either assign a new boundary condition or edit an existing boundary condition to a selected set of boundary nodes or boundary element faces. The dialog that appears depends on whether nodes or faces are selected.

Nodal Boundary Conditions

If a set of nodes is selected when the ***Assign Node/Face BC*** command is selected, the Node BC dialog appears. Boundary conditions assigned to nodes correspond to Dirichlet boundary conditions. Both head and concentration can be specified.

Head

Head boundary conditions in FEMWATER are assigned as total head. FEMWATER converts the total heads to pressure heads internally. Heads can be specified as a constant value or as a transient value (curve of head vs. time).

The Load Data Set button can be used to assign the head values from a data set. For example, if the Elevation data set is selected, the head is set equal to the node elevation at each of the selected nodes.

Concentration

The concentration can also be specified as either a constant or transient value. Since the concentration is a Dirichlet boundary condition, it represents a fixed concentration at the node. It does not represent the concentration of the incoming fluid.

Face Boundary Conditions

If a set of faces is selected when the ***Assign Node/Face BC*** command is selected, the Face BC dialog appears. Boundary conditions assigned to faces are flux-type boundary conditions. Both flux and concentration can be assigned independently. In both cases, the type must be designated as either Variable, Flux (Cauchy), or Flux gradient (van Neumann). The value can be defined as a constant or transient value.

Existing boundary conditions can be deleted by selecting the boundary condition with the Select Boundary Nodes, Select Boundary Faces, or Select Wells tool and selecting the ***Delete BC*** command from the *FEMWATER* menu.

FEMWATER Point / Arc Properties

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Point Properties

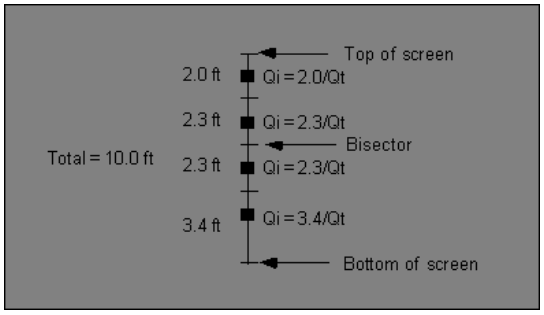
Points in a FEMWATER coverage are used to define injection and extraction wells. The point attributes are edited via the *Point Properties* dialog. The Point Attributes dialog is accessed by selecting a point(s) and then selecting the Attribute Table command in the coverage right click option menu. Each point's properties are displayed in the spreadsheet. The options in the dialog are as follows.

Type - The type of point can be changed to either a well or an observation point. These options are available depending on what was selected in the *Coverage Setup Dialog*. The type for each point is defaulted to "NONE" in the **Type** column pull-down list.

Refine - If the Refine mesh around point option is selected, the edge length of the elements surrounding the node are set to the size entered in the Element size edit field when the 2D projection mesh is generated. (see: Map -> 2D Mesh)

Wells - To define a point as a extraction or injection well change the **Type** to "well" in the spreadsheet. For each well, a flow rate and a concentration can be assigned. Each well is also assigned a the top and bottom elevation of the screened interval.

The screened interval is used to determine which of the nodes in the 3D mesh are used to represent the well in the numerical model. When the ***Map -> FEMWATER*** command is selected, all nodes intercepted by the well screen are found and each node is marked as a point source/sink (a well node). The flow assigned to the well in the conceptual model is distributed to the mesh nodes using the logic illustrated below. A length of influence on the well screen is found for each node and the flux assigned to the node is proportional to the length of influence divided by the total screen length.



The Distribution of Flux Rate to Nodes Overlapped by Well Screen.

Arc Properties

Arcs in a FEMWATER coverage are used to define the model boundary and the boundaries of recharge zones. Arcs on the outside boundary of the model can also be used to specify boundary conditions. Boundary conditions are assigned using the FEMWATER Arc Properties dialog. The Arc Properties dialog is accessed selecting the ***Attribute Table*** command in the coverage right click option menu. The options for arc attributes are as follows:

Flow bc - The Flow bc option is used to define a specified head or specified flux boundary. If this option is selected in the drop-down list. The available options are:

- **Specified head** - If this option is selected, a head value is assigned to each of the two nodes at the endpoints of the arc. If the two values at the endpoints are different, the head is assumed to vary linearly along the arc length. When the ***Map -> FEMWATER*** command is selected, all nodes on the boundary of the mesh beneath the arc are found and the nodes are marked as specified head nodes. A linearly interpolated head value is assigned to each node.
- **Specified flux** - If this option is selected, a flux value is assigned to the arc in the Flux rate column of the spreadsheet. When the ***Map -> FEMWATER*** command is selected, all vertical element faces on the boundary of the mesh beneath the arc are found and the specified flux rate is assigned to the faces.
- **Variable flux** - If this option is selected, a flux value is assigned to the arc. When the ***Map -> FEMWATER*** command is selected, all vertical element faces on the boundary of the mesh beneath the arc are marked as variable boundary faces. If a flux value of zero (the default value) is assigned, the element faces represent a seepage face boundary where below the water table, the head is set equal to the elevation.

Contaminant - The Contaminant option is used to model concentration or mass flux boundary conditions. This option is selected in the Transport bc column in the spreadsheet. The available options are:

Specified concentration - If this option is selected, a concentration value is assigned to the arc using the Concentration flux column. When the ***Map -> FEMWATER*** command is selected, all mesh nodes on the boundary of the mesh beneath the arc are found and the specified concentration is assigned to the nodes.

Specified mass flux - If this option is selected, a mass flux value is assigned to the arc using the Mass flux column. When the ***Map -> FEMWATER*** command is selected, all vertical element faces on boundary of the mesh beneath the arc are found and the specified mass flux rate is assigned to the faces.

Variable (concentration) - If this option is selected, a concentration value is assigned to the arc using the Concentration column. When the ***Map -> FEMWATER*** command is selected, all element faces on the boundary of the mesh beneath the arc are found and the specified concentration is assigned to the faces as a variable type boundary condition. Note that this boundary condition can be used in conjunction with any of the three options for specified head/fluid flux. The proper use of this type of boundary condition is explained in the FEMWATER Reference Manual [1].

Assigning to Zones - The zone column is used to determine how the boundary conditions are applied to the nodes and element faces when the ***Map -> FEMWATER*** command is selected. By default, the boundary conditions are assigned to all nodes and element faces beneath the arc. However, in some cases it is useful to restrict the boundary condition to only a portion of the vertical boundary beneath the arc. This can be accomplished by selecting the Selected zones only option and marking the material zones where the boundary condition is to be applied.

Node Properties - As explained in the previous section, if an arc is marked as a specified head arc, a head value must be assigned to the two nodes at the endpoints of the arc. This is accomplished using the FEMWATER Node Attributes dialog. The Arc Properties dialog is accessed by selecting a node and then selecting the Attribute Table command in the coverage right click option menu. If the head values assigned to the two endpoints of an arc are different, the head is assumed to vary linearly along the length of the arc.

Polygon Properties - Polygons in a FEMWATER coverage serve two purposes: they define the model domain and they can be used to assign recharge values on a zonal basis. When building a FEMWATER coverage, the boundary of the model domain should be delineated using arcs. In order to use the coverage to build a 3D mesh, the arcs should be used to build one or more polygons defining the model domain using the Build Polygon command.

In addition to defining the model domain, a material ID and a recharge value can be assigned to polygons in the FEMWATER coverage using the Polygon Properties dialog. The polygon attributes are as follows:

Material - The Material pull-down list can be used to associate a material with a polygon. When the ***Map -> 2D Mesh*** command is selected, all 2D elements within the polygon are assigned the specified material. This material can be used to define the material type for the 3D elements when the 2D elements are extruded using the ***Fill Between TINs -> 3D Mesh*** command.

Fluid Flux - The Fluid flux option is used to assign a specified flux to the polygon. The flux can be assigned using either the Specified flux or Variable flux options. These optins are set in the *Flow bc* drop-down list. When the ***Map -> FEMWATER*** command is selected, all element faces on the top of the 3D mesh inside the polygon are found and the specified flux is assigned to the element faces.

Contaminant - The Contaminant option is used to assign a mass flux or a concentration. If the Specified mass flux option is selected, the specified mass flux rate is assigned to all element faces on the top of the mesh when the ***Map -> FEMWATER*** command is selected. If the Variable (concentration) option is selected, the specified concentration is assigned to all element faces inside the polygon.

FEMWATER Model Input

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Several model parameters must be defined for a given FEMWATER simulation. In many cases the default values suffice. The following is a list of the required parameters:

Titles

The ***Titles*** command in the *FEMWATER* menu is used to enter two sets of titles. Each set contains three lines of text. The first set is written to the top of the geometry file when the simulation is saved. The second set is written to the top of the model file.

- ***Run Options*** - This command in the *FEMWATER* menu brings up the Run Options dialog. This dialog is used to enter a set of general analysis options.

  Type of Simulation - Three options are available for designating the type of simulations to be performed by FEMWATER:

  Flow only - This option is used to perform a steady state or transient flow simulation.

  Perform a transport simulation only - For this case, a steady state or transient flow simulation must be performed prior to the transport simulation. The results of this simulation (velocity and moisture content) are then input to FEMWATER as a flow solution initial condition.

  Coupled flow and transport - With a coupled flow and transport simulation, either density-dependent flow or density-independent flow can be simulated. This option is controlled by entering the appropriate parameters defining the relationship between concentration and density and concentration and viscosity. These parameters are entered in the *Fluid Properties* dialog.

  Steady State vs. Transient - FEMWATER can be run in either a steady state or transient mode. The steady state mode is only allowed when the Flow only option has been selected.

  Units - The Units button brings up the Units dialog. This dialog is used to enter the units for length, time, concentration, etc. for the simulation. GMS uses the selected unit options to display the appropriate units next to each input edit field in the other FEMWATER dialogs.

  Other Options - The remaining run options are described in the FEMWATER Reference Manual. In most cases, the default values are appropriate.

- FEMWATER Initial Conditions

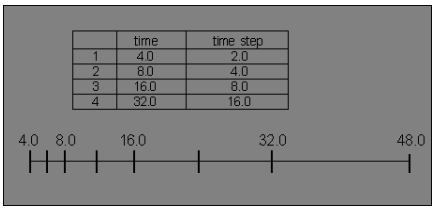
Iteration Parameters

Brings up the Iteration Parameters dialog. This dialog is used to enter the iteration parameters for each simulation type (flow only, transport only, coupled flow and transport).

Particle Tracking Parameters

The Particle Tracking dialog is used to edit parameters relating to how the particle tracking is carried out by FEMWATER during the transport phase.

Time Control Parameters



Time-Line Of Computational Time Steps Defined Using the Variable Time Step Option and the XY Series Editor.

The Time Control dialog is used to enter the data used by FEMWATER to compute the computational time intervals. It is also used to define the reference time. The options in the dialog are as follows:

Time Steps

There are two methods for defining the computational time steps: Constant time step and Variable time step. With the Constant time step method, the first time step is assumed to begin at time 0.0. A constant interval time is entered along with a maximum simulation time. For example, if a constant time step of 2.0 is defined along with a maximum simulation time of 10.0, six computational time steps will be defined at 0.0 (the initial condition), 2.0, 4.0, 6.0, 8.0 and 10.0.

The Variable time step option permits variable intervals between time steps. Selecting the Variable Times button brings up the XY Series Editor. The XY Series Editor has one column for entering times and another for entering time steps. In the time column, the absolute time for a computational time step should be entered. The time step corresponding to each time represents the interval to be used from one specified time to the next.

A simple example of times and time steps defined in the XY Series Editor and the resulting computational time steps to be used by FEMWATER is shown below. In this case the maximum simulation time is equal to 48.0.

Output Control

The Output Control dialog is used to enter parameters defining what type of output will be printed or saved from FEMWATER.

Printed Output File

The left side of the dialog controls what information is written to the printed output file. The printed output file is an ASCII file where the solution will be written.

Data Set Files

The results of a FEMWATER solution are GMS data set files. The Data set files portion of the dialog permits specification of what data sets will be saved and at what frequency. The solution data set files are used as input to GMS to graphically visualize the results. ASCII or binary solution file formats may also be specified. In most circumstances, binary solution files should be specified, since they take up less memory and can be read more quickly by GMS.

Fluid Properties

The ***Fluid Properties*** command in the *FEMWATER* menu brings up the *Fluid Properties dialog*. This dialog is used to specify the acceleration of gravity and the density, viscosity, and compressibility of the fluid.

The **Edit Values** button brings up the *FEMWATER Fluid Coefficients* dialog. This button is undimmed when transport is being analyzed as part of the simulation.

Material Properties

As a 3D finite element mesh is constructed in GMS, a list of materials is defined and each element in the 3D mesh has a material type associated with it. The list of materials is initially created using the *Materials* dialog accessed through the *Edit* menu.

- *Kxx, Kyy, Kzz, Kxy, Kxz, Kyz* - The hydraulic conductivity tensor is defined via the Kxx, Kyy, Kzz, Kxy, Kxz, Kyz fields. Since the tensor is symmetric only the upper right half of the matrix can be specified.

Hydraulic Conductivity Tensor

Kxx	Kxy	Kxz
Kyx	Kyy	Kyz
Kzx	Kzy	Kzz

- *Moisture Content, Relative Conductivity, Water Capacity curves* - These unsaturated zone curves must be defined for each material. The curves can be defined using either the XY Series Editor or the Curve Generator (discussed below). The Curve Generator is accessed via the **Generate Unsat Curves** button in the bottom of the dialog. When that button is selected, the curves that are generated will be associated with the active material, or the material in the spreadsheet row that currently has the focus.

Curve Generator

In most cases, the simplest way to generate a set of pressure head curves for the unsaturated zone is to use the *Curve Generator*. The **Generate Unsat Curves** button brings up the *Curve Generator* dialog. This dialog is used to automatically generate a set of unsaturated zone curves using the van Genuchten equations described in the FEMWATER Reference Manual. The items in the top of the dialog are used to select the curve type ('linear front' or 'van Genuchten equation') and the max height of capillary rise above the water table. Two methods are available for entering the Van Genuchten parameters: (1) you can select the Manual parameter input option and enter the values directly, or (2) you can select the Preset parameter values option and choose from a list of pre-defined soil types.

Once the parameters are defined the **Compute Curves** button can be used to generate a set of curves. The curves are displayed in the bottom of the dialog. New values can be entered and the process can be repeated until a satisfactory result is obtained. When the OK button is selected, the active curves are assigned to the current material.

Each of the unsaturated zone curves is a piece-wise linear curve defined by a sequence of points. The number of points in each curve is either specified by the user or determined automatically by specifying a *Max percent change*. If the **max percent change** option is used, a new point is added to the curve each time the parameter changes by the Max percent value.

Note that the effective porosity for each material is defined from the pressure head vs. moisture content curve. The value at p = 0 is taken from the curve and is written to the model file as part of the MP2 card.

~~Note also that the units of the van Genuchten 'alpha' have been 'hard-wired' into GMS. Because of this the values should always be entered in units of 1/cm. GMS will convert to the model units accordingly.~~Note also that GMS assumes that the van Genuchten 'alpha' value is entered in units consistent with the current model units. GMS displays the current model units next to the input field to help you remember this.

FEMWATER Initial Conditions

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Whenever a FEMWATER analysis is performed, a set of initial conditions must be defined. Initial conditions define the initial status of the pressure head and concentration. Three types of initial conditions are possible for a FEMWATER simulation: cold starts, hot starts, and flow solutions. Cold starts are used to establish a set of initial values at the beginning of a steady state or transient simulation. Hot starts are used to continue a previous run of FEMWATER without having to start over from the beginning. Flow solutions are used to define the flow field that is necessary when performing a transport only simulation (as opposed to coupled flow and transport). Initial conditions are described in more detail in Chapter 7 of the FEMWATER Reference Manual.

Initial conditions are defined using the Initial Conditions dialog. The available options are as follows:

Cold Starts

If a flow only simulation is performed, a set of pressure heads is required for the cold start initial condition. If a transport only simulation is performed, a set of concentrations is required (in addition to the flow solution as explained below). If a coupled flow and transport simulation is being performed, both heads and concentrations are required.

Pressure Head

Two options are available for designating a pressure head cold start initial condition. One option is to enter a constant value into the field labeled Total head. This essentially defines an initial condition corresponding to a flat water table. FEMWATER reads this value and internally generates an array of pressure heads by subtracting the nodal elevations from the given total head value.

The Read from data set file option can be used to designate that the pressure head varies spatially and that the values will be read from a data set file. If this option is selected, the name of the file must be entered at the bottom of the Initial Conditions dialog in the field titled IC pressure head. The data set file is a standard GMS data set file in either the ASCII or binary format. The data set file can be generated using the interpolation options and then saved using the ***Export Data Set*** command accessed by right-clicking on the data set file. However, a simpler approach to generating a well-posed initial condition is to use the **Generate I.C.** button. This button brings up the *Generate Pressure Head Initial Condition* dialog.

The first two items in the *Generate Pressure Head Initial Condition dialog* are used to select a 2D scatter point set and data set. The scatter point set defines a set of elevations corresponding to a best estimate of the final computed water table elevation. A minimum pressure head may also be entered. When the **OK** button is selected, the

elevations in the scatter point set are interpolated to the nodes of the 3D mesh. This defines a total head initial condition. The pressure head initial condition is computed by subtracting the node elevations from the total heads. The user is then prompted for a file name and the pressure head data set is saved to a GMS data set file and the path to the file is automatically written to the IC pressure head field at the bottom of the Initial Conditions dialog.

The pressure head cold start can have a significant influence on the speed of convergence. In some cases, a poorly defined initial condition may even prevent convergence. In most cases, the Read from data set file/Generate IC option should be used since it results in a better initial condition.

Concentration

When defining a set of concentration values for a cold start initial condition, it is often useful to use a constant value of concentration everywhere in the problem domain. For example, in many cases, an initial condition of zero concentration everywhere in the problem domain is appropriate. The Use constant value option can be used to easily define a constant concentration for the entire mesh. If a constant value is not appropriate, the Read from data set file option should be chosen. In this case, the initial condition varies spatially and the values are defined by a data set file. This file can be created by interpolating concentrations to the mesh and saving the resulting data set to a file using the Export command in the Data Tree. When this option is chosen, the name of the data set file containing the concentration initial condition is entered in the IC Concentration field at the bottom of the Initial Conditions dialog.

Hot Start

Hot starts are used to begin a new simulation starting at a given time step of a solution computed from a previous transient simulation. If the hot start option is chosen, FEMWATER reads the specified hot start file and finds the time step corresponding to the specified time. The solution then begins using the data set at that time as the initial condition.

The solution files necessary for a hot start depend on the type of simulation. If a flow only simulation is being performed, a pressure head file is required. If a transport only simulation is being performed, a concentration file is required (in addition to the flow solution described below). If a coupled flow and transport simulation is being performed, both pressure head and concentration are required. When the Hot start option is chosen, the names of the files used for the hot start are entered in the fields at the bottom of the Initial Conditions dialog.

If the Hot start option is chosen, the Append to moisture content file and Append to velocity file options may be selected. If these options are selected, the name of previously computed moisture content and velocity files can be specified in the FEMWATER super file. The file can be edited using a text editor. The super file format is described in the FEMWATER Reference Manual. FEMWATER opens these files and appends the new moisture content and velocity values to the files after the designated hot start time.

Initial Condition File Format

For both the cold and hot start options, data set files can be used to specify the initial conditions. These files can be saved in either ASCII or binary. The format of the files can be specified in the Initial condition file format section. All initial condition files should be either ASCII or binary, i.e., the types cannot be mixed for a given simulation. If the Generate IC button is used to create the pressure head file, the ASCII option should be selected.

Flow Files

A third type of initial condition is required when a transport only simulation is being performed. A transport only simulation utilizes a previously computed flow solution (steady state or transient) to define the three-dimensional flow field required to properly model the contaminant migration. The flow solution consists of a pressure head file.

The flow solution for a transport only simulation is used in combination with either a cold start or a hot start. With a cold start, a set of initial concentration values is provided for concentration in addition to the steady state or transient flow solution. With a hot start, a transient concentration solution and a hot start time is provided in addition to the flow solution.

When a transport only option is selected, the name of the pressure head file from the flow solution is entered at the bottom of the Initial Conditions dialog. The Flow file format options in the Initial Conditions dialog are used to specify whether these files are ASCII or binary, and whether they are steady state or transient.

Saving a FEMWATER Simulation

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Once a FEMWATER simulation has been set up and checked for errors, the final step before running the model is to save the simulation. FEMWATER simulations are saved using the ***Save*** and ***Save As*** commands in the *File* menu.

Selecting the ***Save*** command saves the GMS project including the model simulation. By default the model simulation will be saved to the same location as the GMS project. However, in the Save dialog the path for the model simulation can be specified.

A FEMWATER simulation is actually saved to a set of input files. The FEMWATER super file is a special type of file which is used to organize the set of files used in a simulation. The names of all of the input and output files associated with a simulation are saved in the super file. When FEMWATER is launched, the name of the super file is automatically passed to the FEMWATER executable.

When a FEMWATER simulation is saved, the names of the other FEMWATER input files are automatically patterned after the name of the super file. For example, if the super file is named sampmod.fws, the other files are named sampmod.geo, sampmod.3bc, etc.

Geometry File Options

When a FEMWATER simulation is saved using the ***Save*** or ***Save As*** command, all data associated with the simulation is saved to disk, including both the mesh geometry and the boundary conditions. For large FEMWATER models, the mesh file can take up a substantial amount of disk space and take a long time to save. During a model exercise, it is often the case that multiple versions of the simulation are saved to disk where the only change from one simulation to the next is changes in the analysis options or boundary conditions. In such cases, both time and disk space can be saved by re-using the same geometry file from one run to the next. This can be accomplished by first saving a copy of the FEMWATER simulation to disk and then selecting the ***Geometry File*** command in the *FEMWATER* menu. This command brings up the Geometry File dialog. If the Use existing geometry file option is selected, the Browse button can be used to identify the previously saved geometry file. Once this option is selected, each time GMS saves the FEMWATER super file, it does not re-save the geometry file. Rather, it saves the path to the specified geometry file.

6.1.2. FEMWATER Post-Processing

FEMWATER Display Options

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

The properties of all FEMWATER boundary conditions that GMS displays on the screen can be controlled through the FEMWATER tab of the *Display Options* dialog. This dialog is opened by right clicking on the 3D Mesh Data entry in the Project Explorer, selecting the Display Options command, and then selecting the FEMWATER tab. This tab is only visible when there is a FEMWATER simulation. It can also be accessed from the from the Display menu, the *FEMWATER* menu, or the Display Options macro. The following table describes the display options available for the FEMWATER model.

Display Option	Description
Well	Controls the display of well boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Well (super pump node)	Controls the display of well super pump nodes boundary conditions. The fill color can be adjusted using the button to the left of the display toggle.
Well conc.	Controls the display of well concentration boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Variable	Controls the display of variable boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Variable conc.	Controls the display of variable concentration boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Head	Controls the display of head boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Head conc.	Controls the display of head concentration boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Flux	Controls the display of flux boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Flux conc.	Controls the display of flux concentration boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.

Grad. flux.	Controls the display of gradient flux boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Grad. flux. conc.	Controls the display of gradient flux concentration boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
BC values	Controls the display of boundary condition values. The font, font size, and font color can be adjusted using the button to the left of the display toggle.
Display symbol legend	Controls the display of a legend containing the symbols for the boundary conditions used in the FEMWATER simulation.
Check All	By clicking this button, all of the display options are turned on.
Check None	By clicking this button, all of the display options are turned off.

FEMWATER Post Processing Viewing Options

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Part of the output from FEMWATER is a set of files representing velocity, moisture content, pressure head, and concentration. These files are written in the standard GMS data set file format and can be imported directly to GMS using the ***Read Solution*** command in the *FEMWATER* menu.

Once the FEMWATER solution data sets have been imported to GMS, the standard GMS visualization tools can be used to generate vector plots, cross sections, iso-surfaces, and animations.

One of the output options for FEMWATER is a flux file containing flow budget data for boundary nodes. Once this file is imported, FEMWATER fluxes for selected nodes can be viewed.

Particle Tracking

Particle tracking can be done with a FEMWATER solution, similar to using MODPATH to do particle tracking with a MODFLOW solution. For more information:

- FEMWATER Particle Sets

Flow / Flux

Computed flows can be automatically summed and displayed for FEMWATER simulations. To enable this option, the Save flux file option must be selected in the *FEMWATER Output Control* dialog prior to saving the FEMWATER model. When this option is selected, FEMWATER saves a lumped nodal flow data set file as part of the FEMWATER solution. This file is automatically read into GMS as part of the FEMWATER solution.

Once a FEMWATER flux file has been read into memory as part of a FEMWATER solution, the computed flux through a set of nodes can be displayed simply by selecting the nodes. The flow budget (in, out, net) for the selected nodes is displayed in the **Status Bar** at the bottom of the GMS window. The flow budget is only displayed if the active solution is a FEMWATER solution.

It should be noted that lumped nodal fluxes are only non-zero for boundary nodes where a boundary condition has been assigned.

FEMWATER Particle Sets

FEMWATER
Pre-processing
Building a FEMWATER Model
FEMWATER Model Input
Saving a FEMWATER Simulation
Post-processing
FEMWATER Display Options
FEMWATER Post Processing Viewing Options
Tutorials
FEMWATER Tutorials

Particle tracking can be done with a FEMWATER solution, similar to using MODPATH to do particle tracking with a MODFLOW solution. Particle starting locations can be automatically generated at wells, or at element or element face centroids by using the the commands in the Particle Tracking menu. Once the starting locations are created, the pathlines are automatically computed. Particles are grouped into particle sets, just like with MODPATH.

Restrictions

In order to compute the pathlines, the FEMWATER solution must contain nodal velocity, and nodal moisture content data sets. You can tell FEMWATER to create these data sets using the Output Control dialog. Also, the effective porosity ratio for all materials must be non-zero.

Particle tracking is steady state only. That is, the particles are only influenced by the active time step.

Particles are currently not tracked correctly through hexahedron elements. If you refine the hex elements to create wedges, the particles will be tracked correctly.

Display Options

A set of display options associated with FEMWATER particle tracking is provided in GMS. These options are accessed through the Display Options command in the Particle Tracking menu. This command is only available if a FEMWATER simulation exists. This command brings up the Particles tab of the 3D Mesh Display Options dialog. The following table describes the display options available for the particle tracking.

Display Option	Description
Starting locations	The starting locations for the particles can be turned on and off. The symbol style, color and size can also be adjusted.
Starting location indices	The starting locations indices for the particles can be turned on and off. The font style, color and size can also be adjusted.
Ending locations	The ending locations for the particles can be turned on and off. The symbol style, color and size can also be adjusted.
Pathlines	The pathlines can be toggled on and off, and the pathline style and color can be adjusted. The pathline color can be set to the default color, or the color corresponding to the starting or ending zone codes of the pathlines.
Direction arrows	Direction arrows can be displayed along the pathlines. The size of the arrow heads can be set by specifying the Length in pixels. The arrows are displayed along the pathline at an interval corresponding to the Length specified.
Auto-update	The auto-update can be turned off and on, and controls the automatic updating of the particles.

6.2. MODAEM

MODAEM

MODAEM	
Model Info	
Model type	Analytic element, Steady state analysis
Developer	Vic Kelson of Wittman Hydro Planning Associates
Tutorials	MODAEM Tutorials

MODAEM is an analytic element model developed by Vic Kelson of Wittman Hydro Planning Associates (WHPA) of Bloomington, Indiana. The key feature of analytic element models is that they do not require the discretization of the interior of the model domain into cells and elements as is the case with finite difference and finite element models. Rather, the model is defined by “analytic elements” representing line sources and sinks such as rivers and drains or specified head and specified flow boundaries. Wells are also represented as points, and recharge and aquifer properties can be defined on polygons. MODAEM then formulates a set of equations based on these entities and the equations can be solved for any location in the horizontal plane.

MODAEM is perfectly suited for the conceptual model approach used by GMS since the input data to MODAEM is consistent with the GIS feature objects (points, arcs, and polygons) used to define a GMS conceptual model. In other words, once a MODAEM conceptual model is defined, the model can be immediately executed. There is no need to build a grid or mesh. A background grid is utilized for displaying the MODAEM results using contour lines. However, the background grid is purely for visualization purposes and has nothing to do with the accuracy of the model. MODAEM also supports particle tracking/streamlines.

One of the caveats of the analytical element approach is that a simplified representation of the aquifer must be used. The aquifer properties (K, porosity, bottom elevation, aquifer thickness) can be assigned to polygons, but they are assumed to be constant within each polygon. Currently MODAEM supports steady, confined and unconfined two-dimensional groundwater flow, although streamlines are computed in three dimensions.

Linking MODAEM and MODFLOW

GMS also includes a feature to link MODAEM models with MODFLOW models. The main application of this linkage is for defining boundary conditions for a local scale model. A regional model can be built with MODAEM and then linked to a local scale MODFLOW model. This is accomplished as follows:

1. Build and calibrate the regional scale model using MODFLOW.
2. Build a conceptual model for the local scale MODFLOW model. When setting up the coverage properties for the source/sink coverages, be sure to toggle on the **MODAEM Head** and **Observed Flow** options in the **Source/Sink/BC's** list. If you have more than one MODAEM model in your project, you will also need to select which model should be linked to the MODFLOW model using the **MODAEM models** combo box at the bottom of the *Coverage Setup* dialog.
3. Mark the boundaries (or at least the upstream and downstream boundaries) of the MODFLOW conceptual model as **MODAEM Head boundaries**.

At this point, whenever the ***Map->MODFLOW*** command is executed, GMS will automatically launch the MODAEM model and calculate a head value for each grid cell coinciding with the MODAEM Head boundary.

Furthermore, a flow value is computed for the boundary using MODAEM and assigned to the MODFLOW boundary as an observed head. This makes it possible to monitor the flow budget across the boundary to ensure that the changes made to the local scale model (insertion of remedial wells, etc.) do not violate the flow conditions inherited from the regional scale model.

MODAEM Output File

When MODAEM is executed, it generates an HTML formatted text output file. If the execution is successfully completed, an icon will appear under the corresponding MODAEM conceptual model in the Data Tree representing the output file. Double-clicking on this icon loads the output file into the default web browser.

MODAEM Links

- MODAEM License Agreement
- MODAEM Display Options

MODAEM License Agreement

GNU GENERAL PUBLIC LICENSE

Version 2, June 1991

Copyright (C) 1989, 1991 Free Software Foundation, Inc.

59 Temple Place, Suite 330, Boston, MA 02111-1307 USA

Everyone is permitted to copy and distribute verbatim copies

of this license document, but changing it is not allowed.

Preamble

The licenses for most software are designed to take away your

freedom to share and change it. By contrast, the GNU General Public

License is intended to guarantee your freedom to share and change free

software--to make sure the software is free for all its users. This

General Public License applies to most of the Free Software

Foundation's software and to any other program whose authors commit to

using it. (Some other Free Software Foundation software is covered by

the GNU Library General Public License instead.) You can apply it to

your programs, too.

When we speak of free software, we are referring to freedom, not

price. Our General Public Licenses are designed to make sure that you

have the freedom to distribute copies of free software (and charge for

this service if you wish), that you receive source code or can get it

if you want it, that you can change the software or use pieces of it

in new free programs; and that you know you can do these things.

To protect your rights, we need to make restrictions that forbid

anyone to deny you these rights or to ask you to surrender the rights.

These restrictions translate to certain responsibilities for you if you distribute copies of the software, or if you modify it.

For example, if you distribute copies of such a program, whether gratis or for a fee, you must give the recipients all the rights that you have. You must make sure that they, too, receive or can get the source code. And you must show them these terms so they know their rights.

We protect your rights with two steps: (1) copyright the software, and (2) offer you this license which gives you legal permission to copy, distribute and/or modify the software.

Also, for each author's protection and ours, we want to make certain that everyone understands that there is no warranty for this free software. If the software is modified by someone else and passed on, we want its recipients to know that what they have is not the original, so that any problems introduced by others will not reflect on the original authors' reputations.

Finally, any free program is threatened constantly by software patents. We wish to avoid the danger that redistributors of a free program will individually obtain patent licenses, in effect making the program proprietary. To prevent this, we have made it clear that any patent must be licensed for everyone's free use or not licensed at all.

The precise terms and conditions for copying, distribution and modification follow.

GNU GENERAL PUBLIC LICENSE

TERMS AND CONDITIONS FOR COPYING, DISTRIBUTION AND MODIFICATION

0. This License applies to any program or other work which contains a notice placed by the copyright holder saying it may be distributed under the terms of this General Public License. The "Program", below, refers to any such program or work, and a "work based on the Program" means either the Program or any derivative work under copyright law: that is to say, a work containing the Program or a portion of it, either verbatim or with modifications and/or translated into another language. (Hereinafter, translation is included without limitation in the term "modification".) Each licensee is addressed as "you".

Activities other than copying, distribution and modification are not covered by this License; they are outside its scope. The act of running the Program is not restricted, and the output from the Program is covered only if its contents constitute a work based on the Program (independent of having been made by running the Program).

Whether that is true depends on what the Program does.

1. You may copy and distribute verbatim copies of the Program's source code as you receive it, in any medium, provided that you conspicuously and appropriately publish on each copy an appropriate copyright notice and disclaimer of warranty; keep intact all the notices that refer to this License and to the absence of any warranty; and give any other recipients of the Program a copy of this License along with the Program.

You may charge a fee for the physical act of transferring a copy, and you may at your option offer warranty protection in exchange for a fee.

2. You may modify your copy or copies of the Program or any portion of it, thus forming a work based on the Program, and copy and distribute such modifications or work under the terms of Section 1 above, provided that you also meet all of these conditions:

a) You must cause the modified files to carry prominent notices stating that you changed the files and the date of any change.

b) You must cause any work that you distribute or publish, that in whole or in part contains or is derived from the Program or any part thereof, to be licensed as a whole at no charge to all third parties under the terms of this License.

c) If the modified program normally reads commands interactively when run, you must cause it, when started running for such interactive use in the most ordinary way, to print or display an announcement including an appropriate copyright notice and a notice that there is no warranty (or else, saying that you provide a warranty) and that users may redistribute the program under these conditions, and telling the user how to view a copy of this License. (Exception: if the Program itself is interactive but does not normally print such an announcement, your work based on the Program is not required to print an announcement.)

These requirements apply to the modified work as a whole. If identifiable sections of that work are not derived from the Program, and can be reasonably considered independent and separate works in themselves, then this License, and its terms, do not apply to those sections when you distribute them as separate works. But when you distribute the same sections as part of a whole which is a work based on the Program, the distribution of the whole must be on the terms of this License, whose permissions for other licensees extend to the entire whole, and thus to each and every part regardless of who wrote it.

Thus, it is not the intent of this section to claim rights or contest
your rights to work written entirely by you; rather, the intent is to
exercise the right to control the distribution of derivative or
collective works based on the Program.
In addition, mere aggregation of another work not based on the Program
with the Program (or with a work based on the Program) on a volume of
a storage or distribution medium does not bring the other work under
the scope of this License.
3. You may copy and distribute the Program (or a work based on it,
under Section 2) in object code or executable form under the terms of
Sections 1 and 2 above provided that you also do one of the following:
a) Accompany it with the complete corresponding machine-readable
source code, which must be distributed under the terms of Sections
1 and 2 above on a medium customarily used for software interchange; or,
b) Accompany it with a written offer, valid for at least three
years, to give any third party, for a charge no more than your
cost of physically performing source distribution, a complete
machine-readable copy of the corresponding source code, to be
distributed under the terms of Sections 1 and 2 above on a medium
customarily used for software interchange; or,
c) Accompany it with the information you received as to the offer
to distribute corresponding source code. (This alternative is
allowed only for noncommercial distribution and only if you
received the program in object code or executable form with such
an offer, in accord with Subsection b above.)
The source code for a work means the preferred form of the work for
making modifications to it. For an executable work, complete source
code means all the source code for all modules it contains, plus any
associated interface definition files, plus the scripts used to
control compilation and installation of the executable. However, as a
special exception, the source code distributed need not include
anything that is normally distributed (in either source or binary
form) with the major components (compiler, kernel, and so on) of the
operating system on which the executable runs, unless that component
itself accompanies the executable.
If distribution of executable or object code is made by offering
access to copy from a designated place, then offering equivalent
access to copy the source code from the same place counts as
distribution of the source code, even though third parties are not

compelled to copy the source along with the object code.

4. You may not copy, modify, sublicense, or distribute the Program except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense or distribute the Program is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.

5. You are not required to accept this License, since you have not signed it. However, nothing else grants you permission to modify or distribute the Program or its derivative works. These actions are prohibited by law if you do not accept this License. Therefore, by modifying or distributing the Program (or any work based on the Program), you indicate your acceptance of this License to do so, and all its terms and conditions for copying, distributing or modifying the Program or works based on it.

6. Each time you redistribute the Program (or any work based on the Program), the recipient automatically receives a license from the original licensor to copy, distribute or modify the Program subject to these terms and conditions. You may not impose any further restrictions on the recipients' exercise of the rights granted herein. You are not responsible for enforcing compliance by third parties to this License.

7. If, as a consequence of a court judgment or allegation of patent infringement or for any other reason (not limited to patent issues), conditions are imposed on you (whether by court order, agreement or otherwise) that contradict the conditions of this License, they do not excuse you from the conditions of this License. If you cannot distribute so as to satisfy simultaneously your obligations under this License and any other pertinent obligations, then as a consequence you may not distribute the Program at all. For example, if a patent license would not permit royalty-free redistribution of the Program by all those who receive copies directly or indirectly through you, then the only way you could satisfy both it and this License would be to refrain entirely from distribution of the Program.

If any portion of this section is held invalid or unenforceable under any particular circumstance, the balance of the section is intended to apply and the section as a whole is intended to apply in other circumstances.

It is not the purpose of this section to induce you to infringe any patents or other property right claims or to contest validity of any such claims; this section has the sole purpose of protecting the integrity of the free software distribution system, which is implemented by public license practices. Many people have made generous contributions to the wide range of software distributed through that system in reliance on consistent application of that system; it is up to the author/donor to decide if he or she is willing to distribute software through any other system and a licensee cannot impose that choice.

This section is intended to make thoroughly clear what is believed to be a consequence of the rest of this License.

8. If the distribution and/or use of the Program is restricted in certain countries either by patents or by copyrighted interfaces, the original copyright holder who places the Program under this License may add an explicit geographical distribution limitation excluding those countries, so that distribution is permitted only in or among countries not thus excluded. In such case, this License incorporates the limitation as if written in the body of this License.

9. The Free Software Foundation may publish revised and/or new versions of the General Public License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns.

Each version is given a distinguishing version number. If the Program specifies a version number of this License which applies to it and "any later version", you have the option of following the terms and conditions either of that version or of any later version published by the Free Software Foundation. If the Program does not specify a version number of this License, you may choose any version ever published by the Free Software Foundation.

10. If you wish to incorporate parts of the Program into other free programs whose distribution conditions are different, write to the author to ask for permission. For software which is copyrighted by the Free Software Foundation, write to the Free Software Foundation; we sometimes make exceptions for this. Our decision will be guided by the two goals of preserving the free status of all derivatives of our free software and of promoting the sharing and reuse of software generally.

NO WARRANTY

11. BECAUSE THE PROGRAM IS LICENSED FREE OF CHARGE, THERE IS NO WARRANTY

FOR THE PROGRAM, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE PROGRAM "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE PROGRAM IS WITH YOU. SHOULD THE PROGRAM PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

12. IN NO EVENT UNLESS REQUIRED BY APPLICABLE LAW OR AGREED TO IN WRITING WILL ANY COPYRIGHT HOLDER, OR ANY OTHER PARTY WHO MAY MODIFY AND/OR REDISTRIBUTE THE PROGRAM AS PERMITTED ABOVE, BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE PROGRAM (INCLUDING BUT NOT LIMITED TO LOSS OF DATA OR DATA BEING RENDERED INACCURATE OR LOSSES SUSTAINED BY YOU OR THIRD PARTIES OR A FAILURE OF THE PROGRAM TO OPERATE WITH ANY OTHER PROGRAMS), EVEN IF SUCH HOLDER OR OTHER PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

END OF TERMS AND CONDITIONS

How to Apply These Terms to Your New Programs

If you develop a new program, and you want it to be of the greatest possible use to the public, the best way to achieve this is to make it free software which everyone can redistribute and change under these terms.

To do so, attach the following notices to the program. It is safest to attach them to the start of each source file to most effectively convey the exclusion of warranty; and each file should have at least the "copyright" line and a pointer to where the full notice is found.

}}--> This is free software, and you are welcome to redistribute it
under certain conditions; type `show c' for details.

The hypothetical commands `show w' and `show c' should show the appropriate parts of the General Public License. Of course, the commands you use may be called something other than `show w' and `show c'; they could even be mouse-clicks or menu items--whatever suits your program.

You should also get your employer (if you work as a programmer) or your school, if any, to sign a "copyright disclaimer" for the program, if necessary. Here is a sample; alter the names:

Yoyodyne, Inc., hereby disclaims all copyright interest in the program `Gnomovision' (which makes passes at compilers) written by James Hacker.

, 1 April 1989

Ty Coon, President of Vice

This General Public License does not permit incorporating your program into proprietary programs. If your program is a subroutine library, you may consider it more useful to permit linking proprietary applications with the library. If this is what you want to do, use the GNU Library General Public License instead of this License.

MODAEM Display Options

The properties of all MODAEM properties that GMS displays on the screen can be controlled through the MODAEM tab of the *Display Options* dialog. This dialog is opened by right clicking on the Map Data entry in the Project Explorer, selecting the Display Options command, and then selecting the MODAEM tab. This tab is only visible when there is a MODAEM simulation. It can also be accessed from the from the Display menu, the *MODAEM* menu, or the Display Options macro. The following table describes the display options available for the MODAEM model.

Display Option	Description
Contours	Whenever MODAEM is executed, the head values are computed at the corners of a background grid. These head values can then be contoured. Most of the objects supported by GMS can be contoured by turning on the Contour in the *Display Options* dialog. When an object is contoured, the values associated with the active data set for the object are used to generate the contours.
Map elevations	If the Map Elevations option is selected in the *MODAEM Display Options* dialog, the head values are mapped to elevations (z values) of the background grid and the contours are displayed on the resulting warped grid in oblique view. This makes it possible to see a 3D view of the computed water table surface.
Background mesh	Controls the displays of a background grid. The density of the background grid can be adjusted using the Mesh density factor edit field. The background grid is always adjusted to fit the area defined by the boundary polygon or the GMS graphics window. If you zoom in to look at a local area of the model and select the **F5** button, the background grid is rebuilt to fit the current viewing area. In this manner, one can view the solution at any desired level of detail.
Pathlines	The pathlines can be toggled on and off, and the pathline style and color can be adjusted. The pathline color can be set to the default color, or the color corresponding to the starting or ending zone codes of the pathlines. If the Pathlines option is turned on for one or more MODAEM wells, a set of pathlines are automatically computed by MODAEM and displayed when the solution is updated. For wells, pathlines are always computed backwards in time. The display options associated with the pathlines (max tracking time, direction arrows, etc.) can be controlled using the *MODAEM Display Options* dialog. Particles may also be created at user-specified points in the aquifer, including a starting elevation. GMS will pass these points to MODAEM for forward tracing. Currently, GMS does not provide a way to trace these particles backwards in time.
Direction arrows	Direction arrows can be displayed along the pathlines. The size of the arrow heads can be set by specifying the Length in pixels. The arrows are displayed along the pathline at an interval corresponding to the Length specified.
Dry wells	Controls the display of wells that have gone dry during the MODAEM simulation run. They symbol, symbol size, and symbol color can be adjusted by clicking on the button to the left of the toggle.
Reference point	Controls the display of reference points. They symbol, symbol size, and symbol color can be adjusted by clicking on the button to the left of the toggle.

6.3. MODFLOW

MODFLOW

MODFLOW	
Model Info	
Model type	3D, cell-centered finite-difference, saturated flow model, steady state and transient analysis
Developer	USGS, [1]
Documentation	MF2K Flow Process [2] MF2005 Flow Process [3] MF NWT [4]
Tutorials	MODFLOW Tutorials
Packages	**Flow:** BCF6, HUF, LPF, UPW **Solvers:** DE4, GMG, NWT, PCG2, SAMG, SIP, SOR **Other:** BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR

Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

GMS includes a comprehensive graphical interface to the groundwater model MODFLOW. MODFLOW is a 3D, cell-centered, finite difference, saturated flow model developed by the United States Geological Survey (McDonald & Harbaugh, 1988). MODFLOW can perform both steady state and transient analyses and has a wide variety of boundary conditions and input options.

GMS supports MODFLOW as a pre- and post-processor. The input data for MODFLOW are generated by GMS and saved to a set of files. These files are read by MODFLOW when MODFLOW is launched from the GMS menu. The output from MODFLOW is then imported to GMS for post-processing.

A special version of MODFLOW is distributed with GMS. This version of MODFLOW is the same as the version distributed by the USGS except for a few minor changes primarily related to file input including reading input data from HDF5 files. These changes are clearly marked in the code. For more information see MODFLOW with HDF5.

To learn more about using MODFLOW in GMS, please refer to the available tutorials.

More information about MODFLOW is available from the USGS [1].

Starting from GMS version 8.0, GMS supports MODFLOW 2005.

Starting from GMS version 8.2, GMS supports MODFLOW-NWT.

Pre-Processing

- Importing MODFLOW Files
- Building a MODFLOW Model
- Saving a MODFLOW Simulation
- Parameters
- Stochastic Modeling with MODFLOW
- Interpolate to MODFLOW Layers
- Packages Supported in GMS
- Unsupported MODFLOW Features

Post-Processing

- MODFLOW Display Options
- MODFLOW Post Processing Viewing Options
- Reading a MODFLOW Simulation

MODFLOW Documentation

MODFLOW 88I [5]
MODFLOW 88II [6]
MODFLOW 96I [7]
MODFLOW 96II [8]
MF2K Calibration [9]
MF2K Flow Process [2]

MF2K HUF [10]
MF2K LMG [11]
MF2K LMT [12]
MF2K obssenspeprocess [13]
MF2005 Flow Process [3]
MF NWT [4]

References

[1] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/
[2] http://gmsdocs.aquaveo.com/mf2k_flowprocess.pdf
[3] http://gmsdocs.aquaveo.com/mf2005_flowprocess.pdf
[4] http://gmsdocs.aquaveo.com/MF_NWT.pdf
[5] http://gmsdocs.aquaveo.com/modflow88I.pdf
[6] http://gmsdocs.aquaveo.com/modflow88II.pdf
[7] http://gmsdocs.aquaveo.com/modflow96I.pdf
[8] http://gmsdocs.aquaveo.com/modflow96II.pdf
[9] http://gmsdocs.aquaveo.com/mf2k_calibration.pdf
[10] http://gmsdocs.aquaveo.com/mf2k_huf.pdf
[11] http://gmsdocs.aquaveo.com/mf2k_lmg.pdf
[12] http://gmsdocs.aquaveo.com/mf2k_lmt.pdf
[13] http://gmsdocs.aquaveo.com/mf2k_obssenspeprocess.pdf

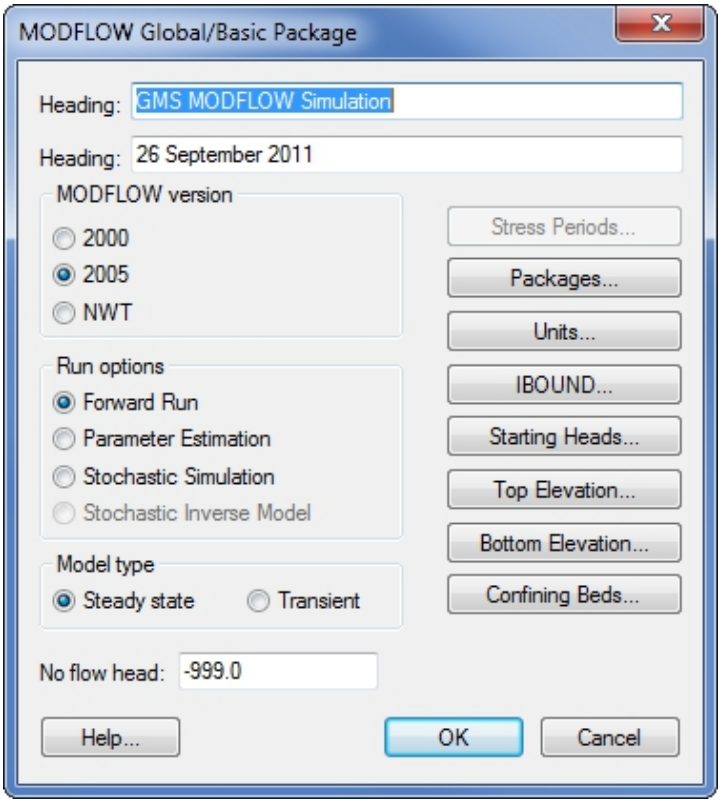
MODFLOW 2005

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Starting at GMS version 8.0, GMS supports MODFLOW 2005.

Choosing MODFLOW 2000 or MODFLOW 2005

Choosing to use MODFLOW 2000 or MODFLOW 2005 is done in the MODFLOW Global Options Dialog. At any time you can switch from one to the other.



MODFLOW Global Option Dialog

Changes in MODFLOW 2005

The following list shows changes in MODFLOW 2005 from MODFLOW 2000.

1. **SOR1 Solver**

- MODFLOW 2005 no longer supports the SOR1 Solver.

2. **PCG2 Solver**

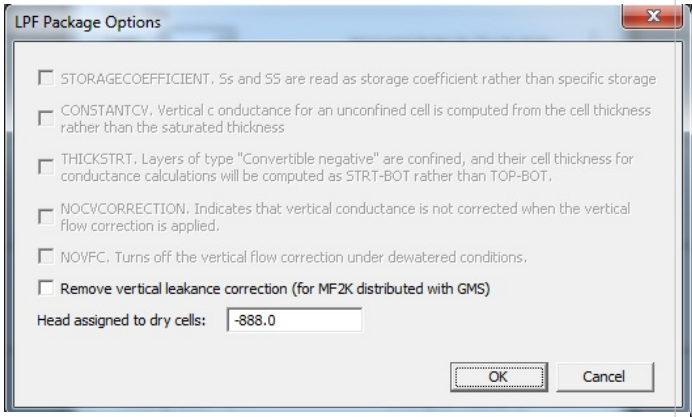
- MODFLOW 2005 replaces the DAMP variable with DAMPPCG and DAMPPCGT variables [1]. These two new variables allow different damping factors to be applied separately to steady-state and transient stress periods.

3. **UZF Package**

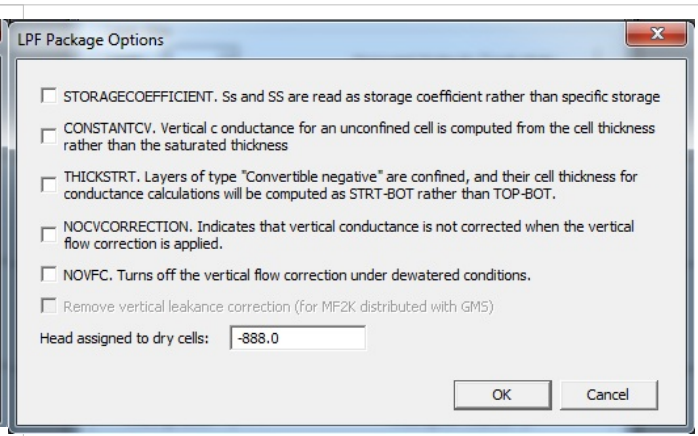
- MODFLOW 2005 added support for the Unsaturated Zone Flow (UZF) package.

4. **LPF package**

- MODFLOW 2005 provides more options for the LPF Package. Pictures below show the LPF package options supported in MODFLOW 2000 vs MODFLOW 2005.



MODFLOW 2000 LPF Package Options



MODFLOW 2005 LPF Package Options

5. **PES and PEST**

- PES is not available in MODFLOW 2005. Automated calibration in GMS is handled using PEST.

References

[1] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/index.html?pcg.htm

6.3.1. MODFLOW Packages

MODFLOW Packages Supported in GMS

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

MODFLOW is divided into a series of components called "packages." Each package performs a specific task. The input for each package is generally contained in a separate file. The MODFLOW packages supported by GMS are listed in the following tables.

The MODFLOW packages to be used in the MODFLOW simulation are specified in the Packages dialog. This dialog is accessed in the *Global Options\Basic Package* dialog. Some of the packages are always required for a simulation and some are optional. One of the flow model packages and one of the solver packages must be selected. Each of the boundary condition/source/sink packages is optional.

For any package not currently supported in GMS, the file associated with that package will be ignored when the model is imported. However, GMS keeps track of unsupported packages it encounters when reading the name file and, when the project is saved, GMS lists the unsupported packages in the name file and copies the unsupported

package files from the old directory to the new directory.

Global Options & Output

Abrev Name	Package/File Name	USGS	Description	Always Req'd?	Extension
MFN	Name file	[1]	The MODFLOW name file. GMS uses "mfn" as the extension.	Yes	*.mfn
BAS6	Basic Package	[2]	Used to specify the grid dimensions, the computational time steps, and an array identifying which packages are to be used.	Yes	*.bas
DIS	Discretization File	[3]	The Discretization File is used to specify certain data used in all models. These include, the number of rows, columns and layers, the cell sizes, the presence of Quasi-3D confining beds, the time discretization.	Yes	*.dis
OUT1	Output Control	[4]	Controls what information is to be output from MODFLOW and when it is to be output.	No	*.oc

Flow Packages

Interfaces to four different flow packages are provided in GMS: the block centered flow (BCF), the layer property flow (LPF), the hydrogeologic unit flow (HUF), and the upstream weighting (UPW) packages. One of these four packages must be selected in the Packages dialog.

Abrev Name	Package Name	USGS	Description	Always Req'd?	Extension
BCF6	Block Centered Flow Package	[5]	Performs the cell by cell flow calculations. The input to this package includes layer types and cell attributes such as storage coefficients and transmissivity. Parameters for sensitivity analysis or parameter estimation are NOT supported.	Yes[1]	*.bcf
LPF	Layer Property Flow Package	[6]	Performs the cell by cell flow calculations. The input to this package includes layer types and cell attributes such as storage coefficients and transmissivity.	Yes[1]	*.lpf
UPW	Upstream Weighting Flow Package	[7]	Based on the LPF package. Modified to provide a linear formulation that can be used with the NWT solver.	Yes[1]	*.upw
HUF	Hydrogeologic Unit Flow Package	[8]	Defines the model stratigraphy in a grid independent fashion (in the vertical direction).	Yes[1]	*.huf

[1] One of the flow packages must be used.

Solver Packages

Abrev Name	Package Name	USGS	Description	Always Req'd?	Extension
GMG	Geometric Multi-Grid	[9]	The Geometric Multigrid Solver is used to solve the finite difference equations in each step of a MODFLOW stress period.	Yes[1]	*.gmg
SIP	Strongly Implicit Procedure	[10]	The Strongly Implicit Procedure package is used to solve the finite difference equations in each step of a MODFLOW stress period.	Yes[1]	*.sip
PCG2	Preconditioned Conjugate Gradient Method	[11]	The Preconditioned Conjugate-Gradient package is used to solve the finite difference equations in each step of a MODFLOW stress period.	Yes[1]	*.pcg
PCGN	Preconditioned Conjugate Gradient Solver with Improved Nonlinear Control	[12]	The Preconditioned Conjugate-Gradient package is used to solve the finite difference equations in each step of a MODFLOW stress period.	Yes[1]	*.pcgn
SOR	Slice-Successive Overrelaxation Method	[13]	The Slice-Successive Overrelaxation package is used to solve the finite difference equations in each step of a MODFLOW stress period.	Yes[1]	*.sor
SAMG or LINK-AMG	Algebraic MultiGrid for Systems Solver	[14]	The Link-AMG package is used to solve the finite difference equations in each step of a MODFLOW stress period.	Yes[1]	*.lmg
DE4	Direct Solver	[15]	The DE4 package is used to solve the finite difference equations in each step of a MODFLOW stress period.	Yes[1]	*.de4
NWT	Newton Solver	[16]	The NWT package is used to solve the finite difference equations in each step of a MODFLOW-NWT stress period.	Yes[1]	*.nwt

[1] One of these solvers must be selected in the Packages dialog.

Once a solver has been selected, the appropriate solver package dialog can be accessed through one of the solver commands (LMG, SIP, PCG2, or SSOR) in the *MODFLOW* menu. The default values shown in each dialog are typically adequate.

For more information on Solver Packages visit: http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/index.html

Optional Packages

Abrev Name	Package Name	USGS	Description	Always Req'd?	Extension
DRN1	Drain Package	[17]	Simulates drain type boundary conditions.	No	*.drn
DRT1	Drain Return Package	[18]	Simulates drain return type boundary conditions.	No	*.drt
EVT1	Evapotranspiration Package	[19]	Simulates the effect of evapotranspiration in the vadose zone.	No	*.evt
ETS1	Evapotranspiration Segments Package	[20]	"allows simulation of evapotranspiration with a user-defined relation between evapotranspiration rate and hydraulic head."[21]	No	*.ets
GAGE	Gage Package	[22]	Allows SFR and lake gaging stations. Gaging station prints time series values for gage location.	No	*.gag
GHB1	General Head Boundary Package	[23]	Simulates a general purpose head-dependent source/sink. Commonly used to simulate lakes.	No	*.ghb

HFB1	Horizontal Flow Barrier Package	[24]	Simulates the effect of horizontal flow barriers such as sheet piles and slurry trenches.	No	*.hfb
LAK3	Lake Package	[25]	A more sophisticated alternative to the typical approach of using the General head package to simulate the effect of lakes and reservoirs. Computes the stage based on the water budget unlike the GHB.	No	*.lak
MNW1	Multi-Node Well 1 Package	[26]	older revision of MNW package used to simulate wells that extend over more than one cell.	No	*.mnw
MNW2	Multi-Node Well 2 Package	[27]	updated revision of MNW1 package used to simulate wells that extend over more than one cell.	No	*.mnw2
RCH1	Recharge Package	[28]	Simulates recharge to the groundwater from precipitation.	No	*.rch
RIV1	River Package	[29]	Simulates river type boundary conditions.	No	*.riv
SFR2	Streamflow-Routing Package	[30]	Simulates the exchange of water between the aquifer and surficial streams. Includes routing and automatic computation of stage. Parameters for sensitivity analysis or parameter estimation are NOT supported.	No	*.sfr
STR1	Stream-Routing Package	[31]	Simulates the exchange of water between the aquifer and surficial streams. Includes routing and automatic computation of stage. Parameters for sensitivity analysis or parameter estimation are NOT supported.	No	*.str
CHD1	Time Variant Specified Head Package	[32]	Simulates specified head boundary conditions where the head is allowed to vary with time.	No	*.chd
SUB1	Subsidence Package	[33]	Simulates simulates aquifer compaction and land subsidence.	No	*.sub
WEL1	Well Package	[34]	Simulates injection/extraction wells.	No	*.wel
UZF1	Unsaturated-Zone Flow Package	[35]	Simulate percolation of water through the unsaturated zone.	No	*.uzf

Other Files

Other files that GMS reads and writes with MODFLOW simulations.

File Name	Description	Extension
MFS	MODFLOW "Super" file, created by GMS to keep track of extra information GMS needs.	*.mfs
MFS	MODFLOW World file, created by GMS to keep track of the geographic location of a MODFLOW model.	*.mfw
PRJ	Projection file in ArcGIS well-known text [36] format specifying a geographic coordinate system.	*.prj
H5	An HDF5 formatted file containing array and list data referenced from other package files. See MODFLOW_with_HDF5 for more information	*.h5
ASP	A file used with a special version of MODFLOW modified to use PEST.	*.asp
PARAM	Extra information about the parameters used by GMS and our modified version of MODFLOW.	*.param
M2P	Used by MF2PEST.EXE program.	*.m2p

Output Files

Standard Extensions for OUTPUT files.

Package	Extension
	*.ccf
Drawdown	*.drw
Global	*.glo
Head	*.hed
Output	*.out

If a fort.# file occurs the MODFLOW file was written to the given unit number.

If a file has an underscore (*_...) followed by an extension refer to pages 29 and 30 of the MODFLOW Documentation linked here:MF2K Obs-Sen-Pes [37]

Notes

[1] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/name_file.htm
[2] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/bas6.htm
[3] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/dis.htm
[4] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/oc.htm
[5] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/bcf.htm
[6] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/lpf.htm
[7] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/upw_upstream_weighting_package.htm
[8] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/huf2.htm
[9] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/gmg.htm
[10] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/sip.htm
[11] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/pcg.htm
[12] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/pcgn.htm
[13] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/sor.htm
[14] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/lmg.htm
[15] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/de4.htm
[16] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/nwt_newton_solver.htm
[17] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/drn.htm
[18] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/drt.htm
[19] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/evt.htm
[20] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/ets.htm
[22] http://water.usgs.gov/nrp/gwsoftware/modflow2000/Guide/index.html?gage.htm
[23] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/ghb.htm
[24] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/hfb6.htm
[25] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/lak.htm
[26] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/mnw.htm
[27] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/mnw2.htm
[28] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/rch.htm
[29] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/riv.htm
[30] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/sfr.htm
[31] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/str.htm
[32] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/chd.htm
[33] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/sub.htm
[34] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/wel.htm
[35] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/index.html?uzf_unsaturated_zone_flow_pack.htm
[36] http://en.wikipedia.org/wiki/Well-known_text
[37] http://pubftp.ems-i.com/download/GMS/GMS%206.5/models/mf2k_obssenspeprocess.pdf

BCF Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

NOTE: The observation, sensitivity, and parameter estimation processes do not support the BCF package.

Once the data in the Global Options/Basic Package are initialized, the data for the flow package can be defined. The BCF package is one of three flow packages that can be used. The BCF package computes the conductance between each of the grid cells and sets up the finite difference equations for the cell to cell flow. It also computes the terms that determine the rate of movement of water to and from storage. The *BCF Package* dialog is accessed through the ***BCF Package*** command in the *MODFLOW* menu.

Data Required

Data arrays required for a particular layer are dependent on the layer type. The layer arrays required for each layer type are shown in the following table. The leakance array is not required for the bottom layer and the storage coefficients are only required for transient simulations.

Layer Type	Required Arrays
Confined (LAYCON=0)	Transmissivity, Leakance, Primary Storage Coefficient (Storativity)
Unconfined (LAYCON=1)	Bottom Elevation, Hydraulic Conductivity, Leakance, Primary Storage Coefficient (Specific yield)
Confined/Unconfined (LAYCON=2)	Top Elevation, Transmissivity, Leakance, Primary Storage Coefficient (Storativity), Secondary Storage Coefficient (Specific yield)
Confined/Unconfined (LAYCON=3)	Top Elevation, Bottom Elevation, Hydraulic Conductivity, Leakance, Primary Storage Coefficient (Storativity), Secondary Storage Coefficient (Specific yield)

Each of the required input arrays must be entered by the user for each layer. Some of the layer arrays can be directly entered. However, some are dependent on the layer geometry. For example, leakance is a function of the layer thickness and the vertical hydraulic conductivity. The transmissivity is equal to the horizontal hydraulic conductivity multiplied by the layer thickness.

The BCF Package can be used for simple models with a single layer for multiple layers with simple stratigraphy. In such cases, many of the parameters are constant for an entire layer and can be entered directly. For more complex models, the following steps can be taken to prepare the input arrays (or the user can use the LPF package):

1. Import a set of scatter points defining the elevations of the stratigraphic horizons.
2. Interpolate the top and bottom elevations of each unit to a 2D grid which matches the 3D computational grid.
3. Compute the desired parameter arrays using the interpolated elevation arrays and the Data Calculator.
4. Copy the parameter arrays into the appropriate MODFLOW arrays in the *BCF Package* dialog.

Since this approach can be quite time-consuming, the LPF Package is recommended for most models.

Layer Data

The aquifer properties are entered in the layer data section.

Layer

The layer edit field is used to select the layer. The buttons on the right side of the dialog apply to the active layer only.

Layer Data Arrays

The six buttons on the right portion of the dialog represent layer data arrays such as elevations and hydraulic conductivity. Each of the six buttons brings up a dialog for entering an array of values. The dialog can be used to edit individual values, assign a constant value to the entire array, or to copy a data set generated by interpolating from a scatter point set to the array.

Not all of the data arrays need to be specified for each layer. Some arrays are only required for transient models. The required arrays depend on the layer type.

Layer data arrays can also be edited using other tools in GMS. The array values can be edited by selecting a set of cells and using the ***Cell Properties*** command. The values in the layer parameter arrays can be initialized using coverages of a conceptual model defined in the Map module. A set of tools for rapidly defining top and bottom elevations is provided (See Defining Layer Elevations). Layer data can also be assigned using material zones.

Layer Type

Each layer must be assigned a layer type (LAYCON). By default, the top layer is defined to be an unconfined layer and all other layers are initialized as confined. The data arrays required by a layer may be dependent on the layer type.

Interblock Transmissivity

The method used for computing interblock transmissivity is specified using the pull-down list in the middle right portion of the *BCF Package* dialog.

Anisotropy Factor

This edit field allows the user to adjust the anisotropy that should be used in the model. This value is defaulted to 1.0.

Head Assigned to Dry Cells

This edit field allows the user to adjust the default head value assigned to dry cells. This value is defaulted to -888.0 or -999.0.

Cell Rewetting Parameters

The controls related to rewetting cells in the lower left portion of the *BCF Package* dialog are only activated if the BCF2 or BCF3 package has been specified. If wetting of cells is to be allowed, a wetting factor, wetting iteration interval, and wetting equation must be specified.

Storage coefficients

Name	Description	Units
Storativity	"the volume of water released from storage per unit decline in hydraulic head in the aquifer, per unit area of the aquifer"[1]	unitless
Specific Storage (Ss)	"the amount of water that a portion of an aquifer releases from storage, per unit mass or volume of aquifer, per unit change in hydraulic head"[1]	1/L
Specific Yield (Sy)	"also known as the drainable porosity, is a ratio, less than or equal to the effective porosity, indicating the volumetric fraction of the bulk aquifer volume that a given aquifer will yield when all the water is allowed to drain out of it under the forces of gravity"[1]	unitless

References

[1] http://en.wikipedia.org/wiki/Specific_storage

CHD Package

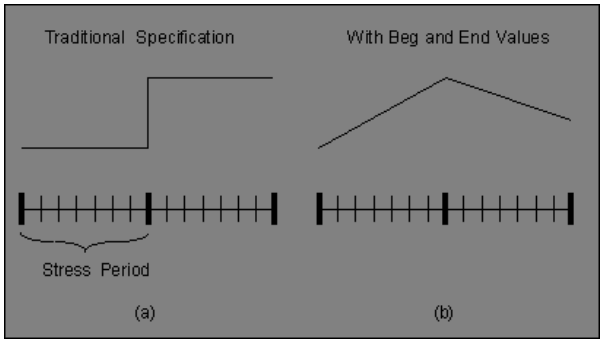
MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

In the original version of MODFLOW, specified head boundaries are defined using a combination of the IBOUND array and the starting heads array. Since both of these arrays are static, boundaries where the head varies with time could not be simulated. To address this type of boundary, a new package called the Time Variant Specified Head (CHD) package has been developed.

NOTE: GMS does not support parameters for the Time Varient Specified Head (CHD) package.

Specifying Transient Head Values

Transient data are handled in a unique fashion with the CHD package. When transient values are assigned to the other stress packages, one value is assigned per stress period. The value represents the value at the beginning of the stress period. This results in a stair step definition of the time series as shown in part a of the following figure. With the CHD package, two values are assigned per stress period: a value at the beginning of the stress period and a value at the end of the stress period. This makes it possible to specify a piece-wise linear time series as shown in part b.



Specifying Transient Values Using (a) the Traditional Approach and (b) the Approach Used in the CHD Package.

Defining Time Variant Specified Head Boundaries

A time variant specified head boundary can be defined using a conceptual model in the Map module simply by using the Transient option when entering the head value in the Attributes dialog. When the specified head attribute is assigned to a polygon, the head is assigned uniformly over the entire polygon. When assigned to an arc, separate head values are applied to each of the nodes on the ends of the arc and the head is assumed to vary linearly between the nodes. When the attribute is assigned to a point, the head is assigned directly to the cell containing the point. For cell-by-cell editing, a selected set of cells can be designated as time variant specified head cells using the ***Point Sources/Sinks*** command described above. When the head vs. time values are entered using the XY Series Editor, two values (beginning and ending values) must be entered per stress period. Once a set of cells has been specified, the Time Variant Specified Head Package dialog can be used to view and edit the values assigned to the cells.

Stress Period

This field shows the current stress period and allows you to cycle through the stress periods and view the different head properties for those stress periods in the spreadsheet below.

Use Previous

Use this option on a stress period to use the values from the previous stress period for the current stress period. This option is unavailable for steady state models and the first stress period of transient models.

Spreadsheet

For cells where CHD type boundary conditions have been assigned, the starting and ending heads assigned to each cell are displayed in the spreadsheet portion at the lower part of the dialog. The spreadsheet can be used to edit the row, column, layer, starting head, and ending head values. For a transient simulation, the values displayed in the spreadsheet are for an individual stress period. The values associated with other stress periods can be edited by entering the number of the desired stress period in the stress period edit box in the center of the dialog. If the Use previous option is selected for a given stress period, the values from the previous stress period are used and the spreadsheet is dimmed.

Reset

This button restores all values in the dialog to their default values

DE4 Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The DE4 package or Direct Solver is one of the solvers available for MODFLOW.

For more information on this solver see the USGS documentation here. [1]

References

[1] http://pubs.er.usgs.gov/usgspubs/ofr/ofr200092

DRN Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Drain package is used to simulate the effect of drains on an aquifer. Drains remove water from the aquifer as long as the water table is above the elevation of the drain. If the water table falls below the elevation of the drain, the drain has no effect. The rate of removal is proportional to the difference in elevation between the water table and the drain. The constant of proportionality is the conductance of the fill material surrounding the drain.

A set of selected cells can be specified as drains using the Point Sources/Sinks command in the MODFLOW menu or drains can be defined using the conceptual model approach in the map module. Drains are specified by assigning an elevation and a conductance to each cell at the location of each drain.

The drain attribute may be associated with points, arcs, or polygons. There are two parameters that are associated with a drain: elevation and conductance. Elevation may be specified as a constant value or transient series, or interpolated from an existing TIN. Conductance may be constant or vary with time.

When a polygon is defined as a drain, the elevation and conductance values (constant or time-varying) are applied uniformly over the entire polygon. When an arc is assigned to be a drain, the conductance is applied uniformly over the arc but separate elevation values are applied to each of the nodes on the ends of the arc and the elevation is assumed to vary linearly between the nodes. When a point is classified as a drain, the elevation and conductance values are assigned directly to the cell containing the point.

See also the Drain Return Package.

Stress Period

This field shows the current stress period and allows you to cycle through the stress periods and view the different river properties for those stress periods in the spreadsheet below.

Use Previous

Use this option on a stress period to use the drain values from the previous stress period for the current stress period. This option is unavailable for steady state models and the first stress period of transient models.

Spreadsheet

For cells where drain type boundary conditions have been assigned, the conductance, elevation, and conductance factor assigned to each cell are displayed in the spreadsheet portion at the lower part of the dialog. The spreadsheet can be used to edit the row, column, layer, conductance, and elevation values. For a transient simulation, the values displayed in the spreadsheet are for an individual stress period. The values associated with other stress periods can be edited by entering the number of the desired stress period in the stress period edit box in the center of the dialog. If the Use previous option is selected for a given stress period, the values from the previous stress period are used and the spreadsheet is dimmed. The conductance factor is used, and only appears in the spreadsheet, with DRN parameter estimation. When doing parameter estimation the value of drain conductance assigned to a particular drain boundary condition will be the value of the drain parameter multiplied by the conductance factor. For more information on DRN parameters and conductance factors see the MODFLOW 2000 manual.

Display Cell IDs/IJK

The radio group at the bottom of the dialog allows the user to adjuster whether the cell ID or the cell i, j, k values are displayed for the cells containing the boundary condition/source/sink.

Delete All

The Delete All button deletes all of the data currently defined in the River package and restores the River package parameters to the default values.

DRT Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Drain Return Package (DRT) is very similar to the regular Drain Package (DRN). Drains remove water from the aquifer as long as the water table is above the elevation of the drain. With the DRT package, some percentage of that removed water can be reintroduced to the aquifer at a specified grid cell.

DRT boundary conditions can be added to selected cells using the ***Point Sources/Sinks*** command in the *MODFLOW* menu. DRT objects can also be defined using the conceptual model approach in the map module.

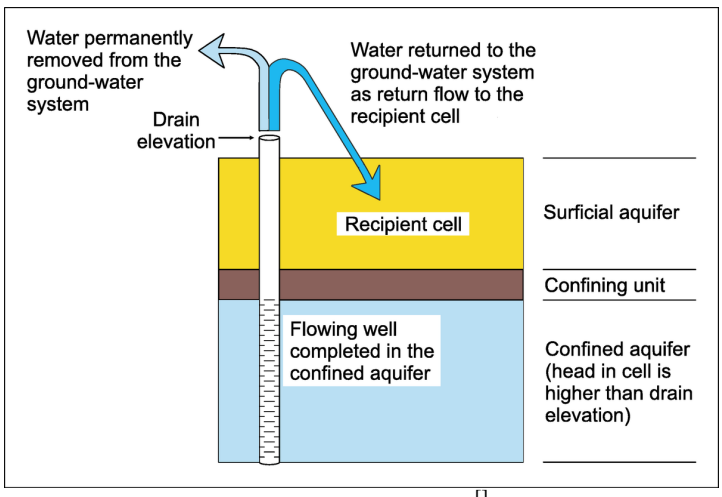


Illustration of DRT package[]

Conceptual model

Drains can be modeled using points, arcs and/or polygons in a conceptual model. The return cell cannot be specified in the conceptual model. This is because the conceptual model is independent of the grid and, in fact, there may be no grid at all when the conceptual model is defined. When GMS performs the ***Map->MODFLOW*** command, the return cell is always set to be the highest active cell above the drain cell. This can later be changed in the Drain package dialog.

The main DRT parameters and how they can be specified are given in the following table.

DRT Input Parameters

Parameter	Points	Nodes	Arcs	Polygons
Conductance	✓		✓	✓
Bottom elevation	✓	✓		✓
Rfprop (return-flow proportion)	✓		✓	✓

When specifying an arc as a DRT arc, the conductance and return flow factor are specified on the arc. Bottom elevation is specified at the nodes on the ends of the arc and is linearly interpolated along the length of the arc.

Care should be taken in entering conductance in the conceptual model so that the units are correct.

Drain Return package dialog

The *Drain Return package* dialog allows the user to edit the location and the input values associated with each drain boundary condition. Drain boundary conditions cannot be created or deleted from this dialog. They can be created and deleted via the ***Map -> MODFLOW*** command, or by using the *MODFLOW Source/Sink* Dialog.

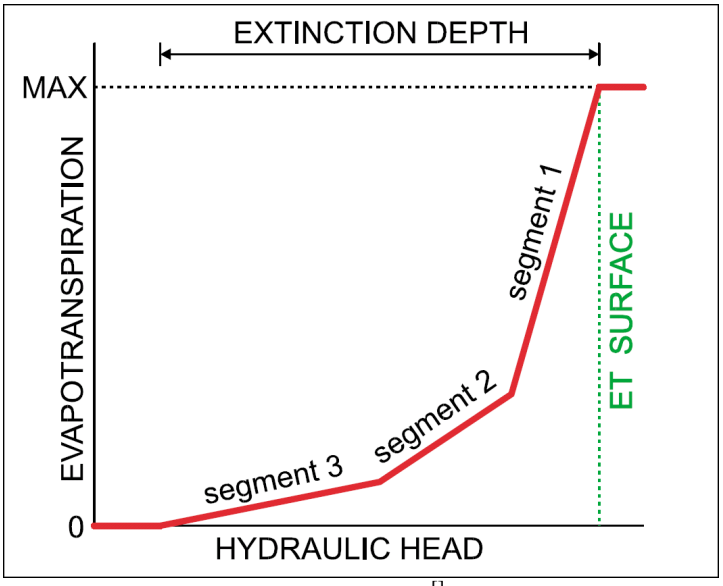
The **Use Return Flow** toggle controls the existence of the "RETURNFLOW" optional keyword in the package input file.

Notes

ETS Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The ETS package is an enhancement of the EVT package and "allows simulation of evapotranspiration with a user-defined relation between evapotranspiration rate and hydraulic head."[] It is included in MODFLOW 2000 and GMS includes an interface to it starting at version 7.0.



From OFR00-466[]

Evapotranspiration Segments

From the package documentation:

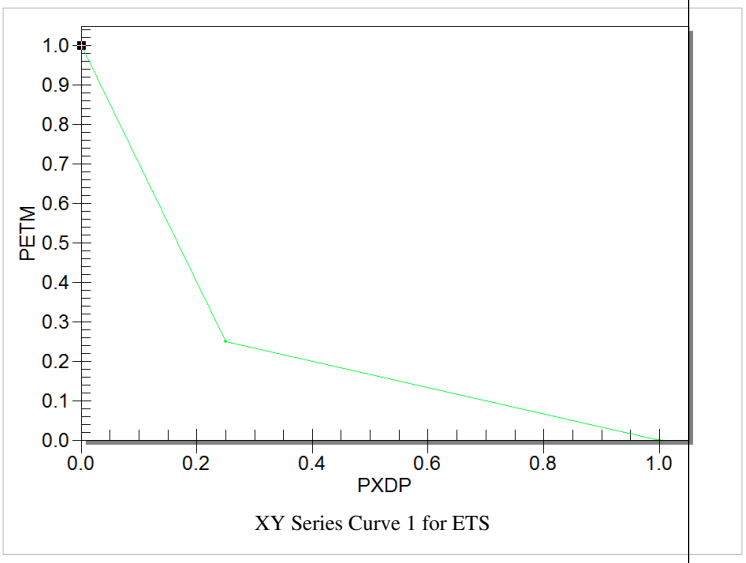
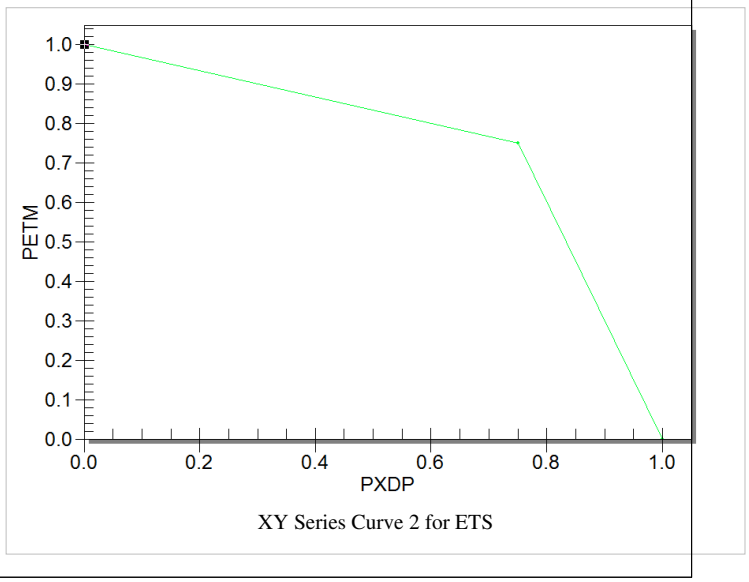
> In the ETS1 Package, the functional relation of evapotranspiration rate to head is conceptualized as a segmented line in the variable interval. The segments that determine the shape of the function in the variable interval are defined by intermediate points where adjacent segments join. The ends of the segments at the top and bottom of the variable interval are defined by the ET surface, the maximum evapotranspiration rate, and the extinction depth. The number of intermediate points that must be defined is one less than the number of segments in the variable interval. For each intermediate point, two values, PXDP and PETM, are entered to define the point. PXDP is a proportion (between zero and one) of the extinction depth, and PETM is a proportion of the maximum evapotranspiration rate. PXDP is 0.0 at the ET surface and is 1.0 at the bottom of the variable interval. PETM is 1.0 at the ET surface and is 0.0 at the bottom of the variable interval. Segments are numbered such that segment one is the segment with its upper endpoint at the ET surface, and segment numbers increase downward. The relation of evapotranspiration rate to head is defined over the model grid by a series of two-dimensional arrays; therefore, PXDP and PETM also are specified as arrays.[]

The input order of the arrays defining the segmented line is important and is defined in the package documentation:

> PXDP-is a proportion of the extinction depth (dimensionless), measured downward from the ET surface, which, with PETM, defines the shape of the relation between the evapotranspiration rate and head. The value of PXDP must be between 0.0 and 1.0, inclusive. Repetitions of PXDP and PETM are read in sequence such that the first occurrence represents the bottom of the first segment, and subsequent repetitions represent the bottom of successively lower segments. Accordingly, PXDP values for later repetitions (representing lower segments) should be greater than PXDP values for earlier repetitions.
>
> PETM-is a proportion of the maximum evapotranspiration rate (dimensionless) which, with PXDP, defines the shape of the relation between the evapotranspiration rate and head. The value of PETM should be between 0.0 and 1.0, inclusive. Repetitions of PXDP and PETM are read in sequence such that the first occurrence represents the bottom of the first segment, and subsequent repetitions represent the bottoms of successively lower segments. Accordingly, PETM values for later repetitions (representing lower segments) generally would be less than PETM values for earlier repetitions.[]

Segmented line functions in GMS

In GMS, the ETS segmented line function can be defined in a conceptual model. When this is done, the standard XY Series Editor is used. The XY Series Editor is a simple tool for creating 2D curves. When used to define the ETS segmented function, the X values correspond to PXDP, and the Y values correspond to PETM. The order of the values is important and is defined by the package documentation as mentioned above. PXDP should be entered in order of increasing value, and PETM should be entered in order of decreasing value. Here are a couple of examples:

Values	Plot
PXDP | **PETM** 0.0 | 1.0 0.25 | 0.25 1.0 | 0.0	 XY Series Curve 1 for ETS
PXDP | **PETM** 0.0 | 1.0 0.25 | 0.25 1.0 | 0.0	 XY Series Curve 2 for ETS

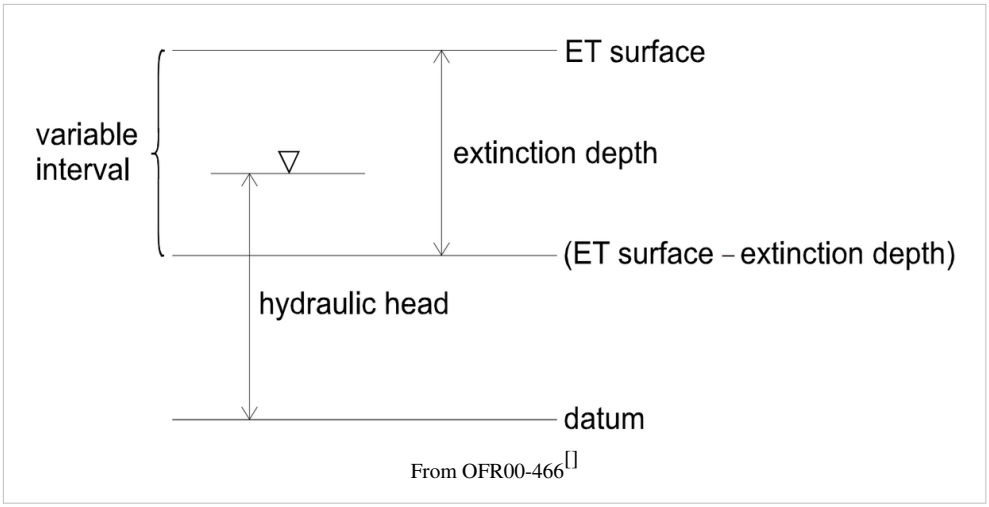
In a GMS conceptual model, it is not possible to define segmented lines which vary with time. However, this can be done in the *ETS Package* dialog.

Notes

EVT Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Evapotranspiration package is used to simulate the effect of plant transpiration and direct evaporation by removing water from cells during a simulation. Evapotranspiration is typically defined by specifying values for each stress period for each vertical column in the grid. The values consist of an elevation, an ET extinction depth, and a maximum ET rate. The elevation is an absolute elevation and the ET extinction depth (measured positive downward) is relative to the specified elevation. If the water table rises above the specified elevation, the evapotranspiration occurs at the maximum ET rate. If the water table falls below the ET extinction depth, evapotranspiration ceases. If the water table elevation lies between these two extremes, the evapotranspiration rate varies linearly with depth. The ETS package, a later extension of the EVT package, allows the rate to vary non-linearly.



From OFR00-466[]

To learn more about Evapotranspiration in MODFLOW go here: http:/ / water. usgs. gov/ nrp/ gwsoftware/ modflow2000/ofr00-466.pdf

EVT Package Dialog

The dialog for editing the evapotranspiration package input data is identical to the Recharge Package dialog except that rather than editing the flux and layer indicator arrays, the evapotranspiration arrays are edited: elevation, ET extinction depth, maximum ET rate, and layer indicator.

Evapotranspiration Option

Two ET options are supported by MODFLOW: ET only at the top layer, and ET at specified vertical cells.

Multiplier

The multiplier is a constant which can be written to the package file with each stress period array. Each value in the array is scaled by the multiplier as the array is imported to MODFLOW. The format button brings up a dialog listing the standard MODFLOW formats. This format is used for displaying the values in the spreadsheet and it controls how the values are written to the package file.

Stress Periods

The values displayed in the spreadsheet are for an individual stress period. The values associated with other stress periods can be edited by entering the ID of the desired stress period in the Stress period edit box in the center of the dialog. If the Use previous option is selected for a given stress period, the values from the previous stress period are used and the spreadsheet is dimmed.

2D Data Set -> Array

The **2D Data Set -> Array** button brings up the Data Browser listing all of the current data sets associated with the current 2D grid. In order for this button to be active, the 2D grid must have the same number of rows and columns as the 3D grid. The selected data set is copied to the recharge array. Data sets are typically generated with the 2D Scatter Point module. The 2D Scatter Point module can be used to interpolate from a scattered set of rainfall measurements to the cell locations. If the data set is transient, the values in the data set are linearly interpolated, if necessary, to each stress period as the data set is copied to the array.

Array -> 2D Data Set

The **Array -> 2D Data Set** button copies the array to the 2D data set list associated with the existing 2D grid.

Constant -> Array

The **Constant -> Array** button brings up a dialog which prompts for a single value. This constant is then assigned to each item in the array for the given stress period.

Spreadsheet

The spreadsheet allows you to edit a 2D array of values, the type depending on the selection in the view/edit combo box. The array types include elevation, et extinction depth, and max ET rate.

Notes

GHB Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The General Head package is similar to the Drain and River packages in that flow in or out of a cell is proportional to a difference in head. General head cells are often used to simulate lakes. General head conditions are specified by assigning a head and a conductance to a selected set of cells. If the water table elevation rises above the specified head, water flows out of the aquifer. If the water table elevation falls below the specified head, water flows into the aquifer. In both cases, the flow rate is proportional to the head difference and the constant of proportionality is the conductance.

A set of selected cells can be specified as general head cells using the ***Point Sources/Sinks*** command in *MODFLOW* menu or general heads can be assigned using the conceptual model in the Map module. When the general head attribute is assigned to a polygons, the head and conductance are applied uniformly over the entire polygon. When assigned to an arc, the conductance is applied uniformly over the arc, but separate head values are applied to each of the nodes on the ends of the arc and the head is assumed to vary linearly between the nodes. When the attribute is assigned to a point, the head and conductance values are assigned directly to the cell containing the point. Once a set of cells have been specified, the *General Head Package* dialog can be used to view and edit the values assigned to the cells.

Stress Period

The stress period field shows the current stress period and allows you to cycle through the stress periods and view the different general head properties for those stress periods in the spreadsheet below.

Use Previous

Use this option on a stress period to use the general head values from the previous stress period for the current stress period. This option is unavailable for steady state models and the first stress period of transient models.

Spreadsheet

For cells where river type boundary conditions have been assigned, the stage, conductance, bottom elevation , and conductance factor assigned to each cell are displayed in the spreadsheet portion at the lower part of the dialog. The spreadsheet can be used to edit the row, column, layer, head, and conductance values. For a transient simulation, the values displayed in the spreadsheet are for an individual stress period. The values associated with other stress periods can be edited by entering the number of the desired stress period in the stress period edit box in the center of the dialog. If the Use previous option is selected for a given stress period, the values from the previous stress period are used and the spreadsheet is dimmed. The conductance factor is used with HFB parameters. When doing parameter estimation the value of head conductance assigned to a particular head boundary condition will be the value of the head parameter multiplied by the conductance factor. For more information on HFB parameters and conductance factors see the MODFLOW 2000 manual.

Display Cell IDs/IJK

The radio group at the bottom of the dialog allows the user to adjuster whether the cell ID or the cell i, j, k values are displayed for the cells containing the boundary condition/source/sink.

Delete All

The **Delete All** button deletes all of the data currently defined in the River package and restores the River package parameters to the default values.

Global Options/Basic Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Once the MODFLOW simulation has been initialized, the next step is to enter the data required by the Global Options/Basic package. This includes data defining fundamental program options such as the computational time intervals (stress periods), an array defining which cells are inactive and which cells have constant heads, an array of starting head values for a transient simulation, and a set of flags defining which of the other packages are to be used. The input data for this package should be entered before editing any of the other packages. The MODFLOW *Global Options\Basic Package* dialog is accessed through the ***Global Options*** command in the *MODFLOW* menu. The options in the dialog are as follows:

Headings

The two headings are optional text strings which are written to the MODFLOW text output file.

Run Options

The run options are not part of the actual MODFLOW input files, but present different ways that MODFLOW 2000 can be run from GMS.

Forward Run

A forward run is a normal MODFLOW simulation. If parameters have been defined for either a stochastic or parameter estimation run, the mean or starting values of the parameters are used for the forward run.

Sensitivity Analysis

The sensitivity analysis option is used to perform a sensitivity analysis on a set of selected parameters. The results can be presented with a set of plots or by contouring data sets representing grid sensitivities. This option is only available with the LPF and HUF flow packages. This option is not available in GMS 6.5 and later (see MODFLOW with HDF5).

After running PEST, sensitivities are displayed in the .sen file. Click here for more information.

Parameter Estimation

In the parameter estimation mode, a set of parameters is found that minimizes the residual between observed and simulated heads and flows. These options are described in more detail on the Automated Parameter Estimation page. This option is only available with the LPF and HUF flow packages.

Stochastic Simulation

The stochastic simulation option is used to perform a stochastic simulation using the Monte Carlo or Latin Hypercube methods for randomizing a selected set of parameters. Stochastic simulations can also be performed using a material set approach (indicator simulations). A risk analysis wizard can be used to analyze the results for a probabilistic threshold analysis or a probabilistic capture zone delineation. These options are described in more detail on the stochastic simulation page.

Stochastic Inverse Model

If the material set option is used for stochastic simulations, the "stochastic inverse" option can be used to perform automated parameter estimation on each of the N candidate model instances in the stochastic. The weighted RMS errors for each of the resulting optimized solutions can be used to weight the computations performed by the risk analysis wizard. This option is only available with the LPF and HUF flow packages.

Model Type

The simulation can be designated as either steady state or transient. If a steady state simulation is specified, certain portions of the MODFLOW interface such as the *Stress Period* dialog are inactivated since they are not relevant.

No Flow Head

This is the value of head to be assigned to all inactive (no flow) cells (IBOUND=0) throughout the simulation.

Stress Periods

The Stress Periods button is used to bring up the *Stress Period* dialog. The stress periods should be defined before any of the sources/sinks are entered. If the steady state option is selected, the **Stress Periods** button is dimmed.

Packages

The Packages button brings up the *Packages* dialog.

Units

The Units button the *Basic Package* dialog brings up the *Units* dialog.

IBOUND

Values

The IBOUND button of the *Global Options\Basic Package* dialog brings up the array editor. The IBOUND array contains a value for each cell in the grid defining the type of the cell as constant head, inactive, or variable head.

Constant Head

A negative value indicates that the cell has a constant head. The value of the constant head is defined in the starting heads array.

Inactive

An IBOUND value of zero indicates that the cell is inactive (no-flow).

Variable Head

A positive IBOUND value indicates that the cell has a variable head (i.e., the head value will be computed as part of the simulation).

Editing the IBOUND Array

There are several ways to change the active/inactive status (positive vs. zero) of a cell before or after initializing the IBOUND array. One method is to directly edit the IBOUND array using the *IBOUND* dialog. Another method is to select the ***Cell Attributes*** command in the *MODFLOW* menu. In most cases, the most efficient method is to use the ***Activate Cells in Coverage*** command in the Map module. This method uses a polygon to define the active and inactive regions.

The constant head cells are typically assigned or edited in one of three ways. One method is to directly edit the IBOUND array. Another method is to select a set of cells and use the *Cell Properties* dialog in the *MODFLOW* menu. The simplest method is to define the constant head zones using feature objects as part of a conceptual model in the Map module.

Starting Heads

The starting head values are used as initial conditions for head for both steady state and transient simulations. The **Starting Heads** button on the left side of the *Basic Package* dialog is used to enter the values of the starting heads array. Selecting the **Starting Heads** button brings up the MODFLOW Array Editor.

Top Elevation

The Top Elevation button brings up an array editor that allows you to edit the top elevation of each layer. GMS requires that the top and bottom elevations for adjacent layers be the same. This means that when you edit the bottom of one layer, move down to the next layer, and edit the top of that layer, the bottom of the previous layer will also be changed. Note that the top and bottom layer elevations must be entered regardless of the which layer data package (BCF, LPF, HUF) is chosen even though these values may not be used because of certain package options.

Bottom Elevation

The Bottom Elevation button brings up an array editor that allows you to edit the bottom elevation of each layer. Note that the top and bottom layer elevations must be entered regardless of the which layer data package (BCF, LPF, HUF) is chosen even though these values may not be used because of certain package options.

Confining Beds

The Layer Confining Beds dialog, accessed from the *Global Options/Basic Package* dialog, allows you to select which layers will have a confining bed below them. The last layer in the grid may not have a confining bed.

Confining beds are most common in older models where the user was trying to conserve memory in solving the groundwater model. In general, it is recommended that you explicitly model all of the layers in the system instead of using confining beds. This feature is supported in GMS to make it possible for users to read in older models.

In the spreadsheet, the user may turn on the confining bed option for any of the layers except the last layer in the grid.

The Confining Bed Elevations button brings up the array editor that allows the user to edit the elevation of each layer.

GMG Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The GMG package is one of four solver packages for MODFLOW supported by GMS. The GMG solver is an iterative solver based on a multi-grid approach.

For more info on the GMG solver look at the following document: MODFLOW-2000, THE U.S. GEOLOGICAL SURVEY MODULAR GROUND-WATER MODEL -- GMG LINEAR EQUATION SOLVER PACKAGE DOCUMENTATION By: J. D. Wilson, NRC Post-Doc, and R. L. Naff, U.S. Geological Survey [1]

References

[1] http://pubs.usgs.gov/of/2004/1261/

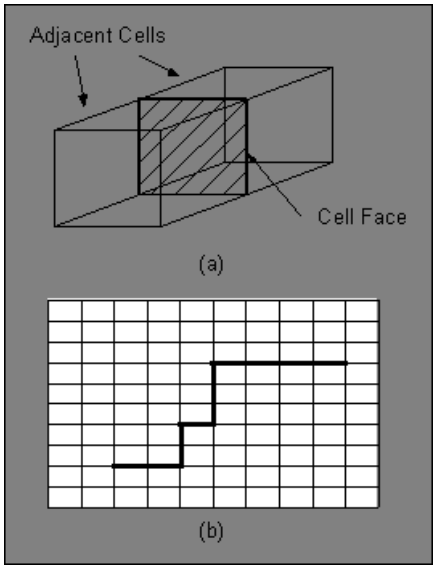
HFB Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Horizontal Flow Barrier (HFB) package is used to simulate the effect of sheet pile walls, slurry trenches, or other objects which act as a barrier (or partial barrier) to horizontal flow. Barriers are simulated in the HFB package by identifying cell boundaries which approximately coincide with the location of the barrier and assigning a hydraulic characteristic to each cell boundary. Each cell boundary represents a vertical face between two adjacent cells as shown in the following figure.

Hydraulic Characteristic

The original version of the HFB Package required the input of the hydraulic characteristic either as barrier transmissivity divided by the width of the horizontal-flow barrier (for layer types 0 and 2 in BCF) or as barrier hydraulic conductivity divided by the width of the horizontal-flow barrier (for layer types 1 and 3 in BCF). In the current HFB Package, the hydraulic characteristic is always the barrier hydraulic conductivity divided by the width of the barrier, regardless of the layer type or flow package (BCF or LPF) used; thus, layer thickness is always used in calculating the contribution to the conductance terms. The HFB Package uses cell elevations specified in the discretization file to calculate cell thickness. Cell thickness is head dependent for layer types 1 and 3 in the BCF Package and for convertible layers in the Layer-Property Flow Package. (taken from MODFLOW 2000 documentation)



Cell Boundary Used to Represent Horizontal Flow Barrier. (a) Oblique View, (b) Plan View.

Defining Barriers

Barriers are defined in one of two ways: (1) they can be defined using a set of arcs in the Map module or (2) they can be defined one cell boundary at a time using the ***Toggle Barrier*** command in the 3D Grid module.

Using the Map Module

In most cases, the simplest method is to create one or more Horizontal Flow Barrier arcs in the Map module corresponding to the barriers and let GMS automatically find the closest sequence of cell boundaries and mark them as barriers.

Using the Toggle Barrier Command

Horizontal flow barriers can also be defined one at a time by selecting two adjacent cells and selecting the ***MODFLOW\Toggle Barrier*** menu command. This brings up a dialog that can be used to mark the boundary between the two selected cells as a barrier and to enter a hydraulic characteristic for the barrier. This same command can be used to delete a barrier between two cells.

HFB Package Dialog

Regardless of which method is used to define the barriers, an existing set of barriers can be viewed and edited using the ***HFB Package*** command in the *MODFLOW* menu. This command brings up the HFB Package dialog. This dialog can be used to edit the location and hydraulic characteristic of each of the currently defined barriers.

Spreadsheet

For cells where horizontal flow barriers have been assigned, the hydraulic characteristic assigned to each cell is displayed in the spreadsheet portion at the lower part of the dialog.

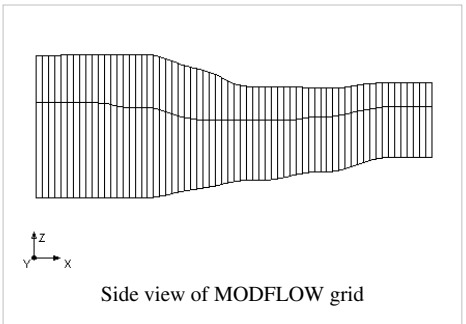
Reset

The **Reset** button can be used to delete all barriers.

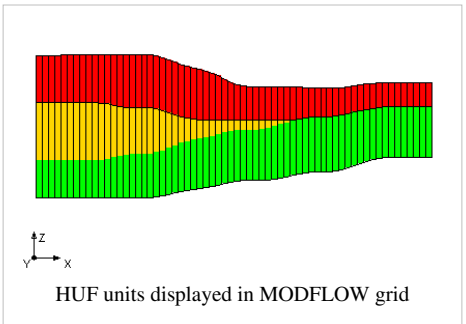
HUF Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

This package is an alternative to the BCF and LPF packages. The HUF package allows the user to define the vertical elevations of the hydrogeology independent of the 3D grid as shown in the figures below. Notice in the figures below that the second grid layer has more than one hydrogeologic unit associated with it.



Side view of MODFLOW grid



HUF units displayed in MODFLOW grid

With the HUF package, the user then defines the top and thickness of each of the hydrogeologic units. A material is associated with each unit and the hydraulic properties (HK, VK...) are assigned to the material. The top and thickness

arrays can be automatically generated from solids using the ***Solids -> HUF*** command from the *Solids* menu or from boreholes with the ***Horizons -> HUF*** command in the *Boreholes* menu.

Another feature in GMS that incorporates the HUF package is the ability to use TPROGS to Generate HUF Data.

Once the HUF package has been chosen using the packages dialog, the HUF package dialog can be accessed through the ***HUF package*** command in the *MODFLOW* menu. The following options are available in the HUF Package dialog.

Layer

This field displays the current layer. Selecting any of the input array buttons will display the corresponding layer initially. For multiple layer models, this field can be incremented or decremented to show other layers.

Layer type

This radio group selects the layer type for the current layer. By default, all layers are convertible.

Hydrogeologic Unit Properties

The window lists the names of the currently defined units. The Top and Thickness button bring up the MODFLOW Array Editor. For each hydrogeologic unit the user can specify if vertical hydraulic conductivity (VK) or vertical anisotropy (VANI) will be used. The material selection box show the material the is assigned to the unit. The material properties (HK, VK...) are edited by selecting the **Edit Materials** button.

Vertical hydraulic conductivity

The HUF package has the option to enter vertical hydraulic conductivity values as either actual hydraulic conductivity values or as anisotropy factors dependant on horizontal hydraulic conductivity. These options are only available for multi-layer models.

Head Assigned to Dry Cells

This is the value that MODFLOW will give to cells with a head level below the bottom elevation of the cell (dry cell). When reading in a solution, GMS will mark cells with this value with a dry cell symbol.

Cell Rewetting Parameters

If wetting of cells is allowed, a wetting factor, wetting iteration interval, and wetting equation must be specified.

LAK Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Lake Package is a more sophisticated alternative to the typical approach of using the General Head Package to simulate the effect of lakes and reservoirs on an aquifer. Comparing the Lake package to the GHB Package is similar to comparing the River Package to the Stream Package. With the GHB Package, the user defines the stage. With the Lake package, the stage is computed automatically based on the water budget. The water budget is a function of inflow, outflow, recharge, evaporation, etc. The storage capacity of the lake is determined automatically based on the lake geometry.Some of the cells defining the lake are allowed to go dry if the lake stage drops sufficiently.

The lake is also assumed to have lakebed sediments which affect the flow between the aquifer and the lake. The effect of the lakebed sediments is represented with a leakance term which includes the thickness and the hydraulic conductivity.

The Lake Package includes an option that allows one to simulate "sublakes". This option makes it possible for the lake to partition into smaller, separated lakes as the lake stage drops.

The Theta term offers the user a choice between an explicit solution at 0.0, a semi-implicit solution between 0.0 and 1.0, or an implicit solution at 1.0

Terms

- Leakance - Represents the lakebed leakance. This is a function of hydraulic conductivity and lakebed thickness
- Bathymetry - A reference to a TIN from which the bathymetric elevations are derived.
- Sill Elev - The sill elevation at which the sublake begins to drain into the parent lake.
- Withdrawal - The volumetric rate, or flux, of water removal from a lake by means other than rainfall, evaporation, surface outflow, or ground-water seepage.

LMG Package

The LMG1 package is only supported in GMS versions prior to 8.0. The LMG1 Package information can be found here.

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

LMG3 package offers some improvements over the LMG1 package for certain types of problems. The LMG3 package is backward compatible with LMG1 input files. For non-linear problems and simulations which use several time steps, subsequent matrices to be solved for often change only slightly in their algebraic properties. In this case the LMG3 offers a possibility to accelerate the overall simulation time enormously.

MXITER. Maximum outer iterations

This value is the maximum iterations, or calls to the solver. For linear problems, this value can be 2. For non linear problems, this values should be larger, but generally less than 50.

MXCYC. Maximum inner iterations

This value is the maximum number of iterations per call to the solver. The default value is 50, and this is sufficient for most problems.

RCLOSE. Inner iteration closure criterion

Same as RCLOSE for the GMG solver package. The PCG algorithm computes the l2norm of the residual and compares it against RCLOSE. In LMG1 this was called Budget Closure Criterion. When the scaled norm of the matrix equations is less than this value then the solver iteration stops.

HCLOSE. Outer iteration closure criterion

Head change convergence criterion for nonlinear problems. After each linear solve (inner iteration), the maximum norm of the head change is compared against HCLOSE.

CONTROL. LMG version to use

This variable allows the user to select either LMG1 or LMG2 version of the LMG solver.

Damping method

Three different damping methods are available: constant, Adaptive damping using the Cooley method, or Adaptive RRR damping using the relative reduced residual. Refer to the official package documentation for more explanation on the different damping methods.

DAMP. Accleration Parameter

This value is used if the acceleration method is set to constant. Usually, this value is 1.0, but for some problems lower values may help achieve convergence.

DUP. Max Acceleration

This is the max damping value when using the adaptive damping with relative reduced residual.

DLOW. Min Acceleration

This is the min damping value when using the adaptive damping with relative reduced residual.

Conjugate Gradient

Controls whether the conjugate gradient method is used to accelerate the solver. By default this option is on.

Stor1, Stor2, Stor3

These values are ignored in LMG2. In LMG1, these variables are used to control the amount of memory storage used by the solver. The defaults rarely need to be changed.

IOUTAMG. Output control

This controls the information printed to the samg output file while the solver is running.

Reset

This button will change all options to the default state.

LMG1 Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The LMG package is one of four solver packages for MODFLOW supported by GMS.

Maximum outer iterations

This value is the maximum iterations, or calls to the solver. For linear problems, this value can be 2. For non linear problems, this values should be larger, but generally less than 50.

Maximum inner iterations

This value is the maximum number of iterations per call to the solver. The default value is 50, and this is sufficient for most problems.

Acceleration type

Three different acceleration methods are available: constant, adaptive damping using the Cooley method, or adaptive damping using the relative reduced residual. Refer to the official package documentation for more explanation on the different damping methods.

Accleration Parameter

This value is used if the acceleration method is set to constant. Usually, this value is 1.0, but for some problems lower values may help achieve convergence.

Max Acceleration

This is the max damping value when using the adaptive damping with relative reduced residual.

Min Acceleration

This is the min damping value when using the adaptive damping with relative reduced residual.

Budget Closure Criterion

When the scaled norm of the matrix equations is less than this value then the solver iteration stops.

Conjugate Gradient

Controls whether the conjugate gradient method is used to accelerate the solver. By default this option is on.

Stor1, Stor2, Stor3

These variables are used to control the amount of memory storage used by the solver. The defaults rarely need to be changed.

Reset

This button will change all options to the default state.

LPF Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

This package is an alternative to the BCF and HUF packages and is similar to the "true layer" option used with the BCF package in version 3.1. With MODFLOW 2000, the layer elevations (top and bottom) are defined as input to the Global Process (using the Global Options Dialog), regardless of which flow package is being used. With the LPF package, the user then defines the horizontal and vertical hydraulic conductivity for each layer. MODFLOW then computes the cell by cell conductances using the K values and the layer geometry.

Other noteworthy features include the ability to enter horizontal anisotropy values on a cell by cell basis. There is also an option to specify vertical anisotropy factors rather than vertical hydraulic conductivity values. This option is particularly useful when performing automated parameter estimation since it ties the K_v to K_h and eliminates the need to define K_v as an independent parameter.

Another feature of the LPF package relative to the BCF package is that there are now only two layer types: confined and convertible. A convertible layer is similar to the LAYCODE = 2 and LAYCODE = 3 types in the BCF package. The layer can be either confined or unconfined depending on the elevation of the computed water table.

Once the LPF package has been chosen using the packages dialog, the LPF package dialog can be accessed through the LPF package command in the *MODFLOW* menu.

Use data arrays

This option allows you to enter conductivities on a layer-by-layer basis. When this option is chosen, the Horizontal Hyd. Conductivity…, Vertical Hyd. Conductivity, Specific Storage, etc. buttons will be available.

Use material ids

This options uses material sets in the place of property arrays. Property values are entered on a material-by-material basis. When MODFLOW files are saved, GMS internally replaces the correct values from the materials to the property arrays. When this option is chose, the property array buttons are replaced with the **Material Properties** and **Material IDs** buttons.

Layer

This field displays the current layer. Selecting any of the input array buttons will display the corresponding layer initially. For multiple layer models, this field can be incremented or decremented to show other layers.

Layer type

This radio group selects the layer type for the current layer. By default, all layers are convertible. Here you can change the layer type to confined.

Vertical hydraulic conductivity

The LPF package has the option to enter vertical hydraulic conductivity values as either actual hydraulic conductivity values or as anisotropy factors dependant on horizontal hydraulic conductivity. Vertical anisotropy (VKA) "is the ratio of horizontal to vertical hydraulic conductivity. In this case, HK is divided by VKA to obtain vertical hydraulic conductivity, and values of VKA typically are greater than or equal to 1.0."[1] These options are only available for multi-layer models.

Interblock transmissivity

The method used for computing interblock transmissivity is specified using the pull-down list in the middle right portion of the *LPF Package* dialog.

Layer Data Arrays

The eight buttons in the lower right portion of the dialog represent layer data arrays such as elevations and hydraulic conductivity. Each of the eight buttons brings up the MODFLOW Array Editor. Not all of the data arrays need to be specified for each layer. Some arrays such as Specific Storage and Specific Yield are only required for transient models.

Head Assigned to Dry Cells

This is the value that MODFLOW will give to cells with a head level below the bottom elevation of the cell (dry cell). When reading in a solution, GMS will mark cells with this value with a dry cell symbol.

Remove vertical leakance correction

This is the value that MODFLOW will give to cells with a head level below the bottom elevation of the cell (dry cell). When reading in a solution, GMS will mark cells with this value with a dry cell symbol. The Layer Property Flow Package of MODFLOW-2000 was modified by A.W. Harbaugh (U.S. Geological Survey, written commun., 2002) to remove the vertical leakage correction for conditions in which a partially saturated cell is immediately below a fully or partially saturated cell (Harbaugh and others, 2000, p. 31-33). The vertical leakage correction

simulates perched conditions within an aquifer system. However, perched conditions are not known to be widespread on a basin scale. The vertical leakage correction adds an additional nonlinear term to the model (A.W. Harbaugh, U.S. Geological Survey, written commun., 2002), which resulted in several of the numerical solvers (SIP (strongly implicit procedures), SOR (slice-successive overrelaxation), and PCG (Hill, 1990; Harbaugh and others, 2000) not coming to a solution and the LMG (link-algebraic multigrid; Mehl and Hill,2001) solver providing unacceptable volumetric budget errors (wrir02-4200.pdf) . Thus in the LPF package the user has the option to turn off the vertical leakance correction to account for these factors. As the user turns on the "Remove Vertical Leakance Correction" toggle, GMS then uses the NOCVCORRCTION [2] option that is found in MODFLOW2005 to remove vertical leakance corrections.

Cell Rewetting Parameters

If wetting of cells is allowed, a wetting factor, wetting iteration interval, and wetting equation must be specified.

Notes

[2] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/index.html?lpf.htm

MNW1 Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Multi-Node Well (MNW1) package was developed to more accurately model wells that are completed in multiple aquifers or in a single heterogeneous aquifer, partially penetrating wells, and horizontal wells that can be affected by the effects of dynamic changes in the distribution of pumping or intraborehole flow that can significantly alter groundwater flow. A more recent revision of the MNW package is available in the MNW2 package. A MODFLOW model can not use both the MNW1 and MNW2 packages, it must use one or the other.

MNW1 boundary conditions can be added to selected cells using the ***Point Sources/Sinks*** command in the *MODFLOW* menu. MNW1 points can also be defined using the conceptual model approach in the map module.

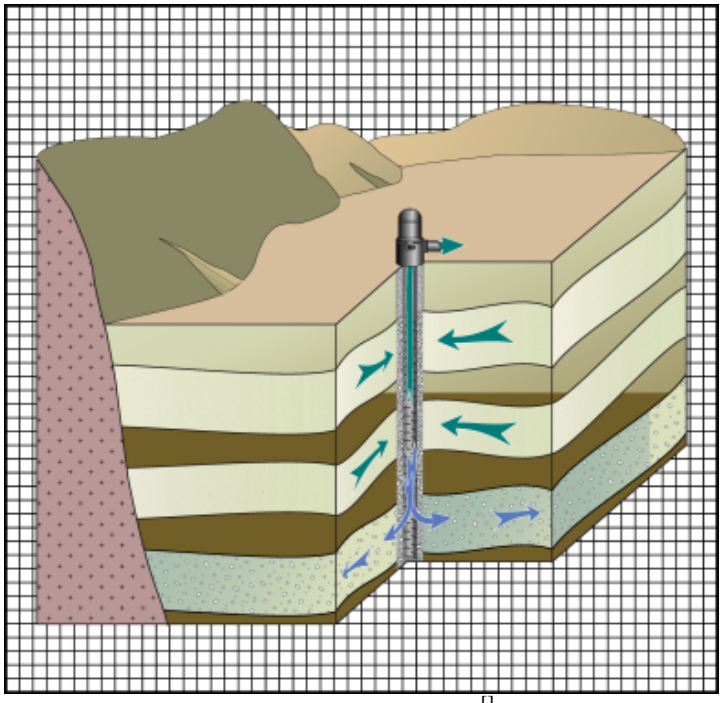


Illustration of MNW package[]

Conceptual model

MNW1 Wells can be modeled using points in a conceptual model. The user can specify the various MNW1 properties associated with points in a coverage. In addition, a multi-node MNW1 boundary condition (bc) can be created using the conceptual model approach. The easiest way to create a multi-node MNW1 bc is to define a well screen associated with the MNW1 point that crosses multiple layers of the MODFLOW grid. Another method for creating a multi-node MNW1 bc is to specify a common "Well ID" to group different points together. Thus, multiple points in the same or different coverages can be used to create multi-node MNW1 bcs.

Multi-Node Well package dialog

The Multi-Node Well (MNW1) package dialog allows the user to edit the location and the input values associated with each mnw boundary condition. MNW1 boundary conditions cannot be created or deleted from this dialog. They can be created and deleted via the ***Map -> MODFLOW*** command, or by using the *MODFLOW Source/Sink* Dialog.

The upper section of the dialog controls global options for the package. The spreadsheet in the lower section of the dialog controls the bcs that vary with each stress period. The user is referred to the MNW1 documentation for a full explanation of each of the inputs to the MNW1 boundary conditions. In GMS we have added the Name, Active, and Well ID fields. Name is a label used to identify a boundary condition. Active is a flag used to determine if the boundary condition is used during a particular stress period. The Well ID field is used to identify multi-node wells.

Notes

MNW2 Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Multi-Node Well (MNW2) package is an updated revision of the MNW package meant to replace the MNW1 Package. It includes modifications to better model partially penetrating wells, non-vertical wells, and includes the ability to specify a pump performance curve. The MNW package was developed to more accurately model wells spanning multiple aquifers or in a single heterogeneous aquifer, partially penetrating wells, and horizontal wells that can be affected by the effects of dynamic changes in the distribution of pumping or intraborehole flow that can significantly alter groundwater flow. A MODFLOW model can not use both the MNW1 and MNW2 packages, it must use one or the other. A detailed description of the MNW2 input is available at the USGS website [27].

MNW2 boundary conditions can be directly added via the Multi-Node well (MNW2) package dialog. Boundary conditions can also be added to selected cells using the ***Point Sources/Sinks*** command in the *MODFLOW* menu. For a conceptual model MNW2 points can be defined in the map module.

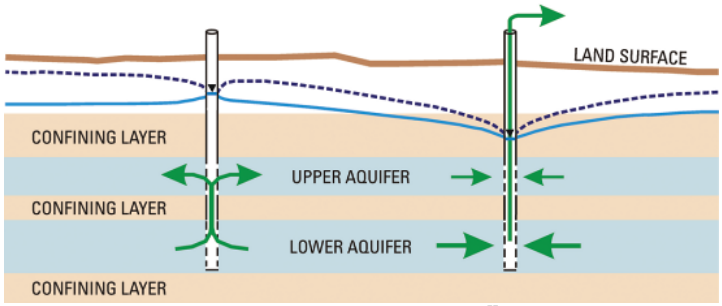


Illustration of MNW2 package[]

Conceptual Model

An MNW2 well can be added to a conceptual model by creating an MNW2 point. The geometry of the well screen can be set using the **Use screen** attribute for a single screen, or for multiple screens the geometry can be set using the **Boreline** attribute button. If niether one is set, the generated well nodes include the grid layers specified for the coverage by its default layer range.

Multi-Node Well (MNW2) Package Dialog

MNW2 Wells can be edited in the *Multi-Node Well (MNW2) Package* Dialog. The main dialog allows editing stress period data for wells. The stress period field shows the current stress period and allows you to cycle through stress periods and view the well properties for each stress period in the spreadsheet below. MNW2 AUX fields can be added and removed by clicking on the **Edit AUX...** button. The values are displayed in the spreadsheet. Controls to edit the print detail (MNWPRNT) and the MNWI package values are available below the spreadsheet. Wells can be added, edited and removed by clicking on the **Wells...** button to bring up the MNW2 Wells Dialog.

MNW2 Wells Dialog

The *MNW2 Wells* Dialog allows wells to be added, removed, and edited. It consists of two spreadsheets. The top spreadsheet lists the table of MNW2 wells. When a well in the top spreadsheet is clicked on, the nodes for the well are listed in the bottom spreadsheet. Nodes can be added and removed by changing the NNODES column in the wells spreadsheet.

Sources/Sinks Dialog

An MNW2 well can be added or edited for a selected cell using the *Sources/Sinks* menu item under the *MODFLOW* menu. The well info can be edited from the Sources/Sinks Dialog by clicking on the button in the Edit column.

Notes

NWT Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Newton package (NWT) is one of the solvers available in MODFLOW-NWT. The NWT package is only used with the UPW Package. Using the NWT solver requires twice the memory of other MODFLOW solvers because using the Newton method results in an asymmetric matrix. Because of the asymmetric matrix a different matrix solver must be used. The NWT package provides an option to use one of two matrix solvers: a generalized-minimum-residual (GMRES) solver and an Orthomin/stabilized conjugate-gradient solver called *x*MD (chi-MD).

For more information on the NWT package see the USGS documentation here. [16]

PCG2 Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The PCG2 package is one of four solver packages for MODFLOW supported by GMS.

Maximum outer iterations

This value is the maximum iterations, or calls to the solver. For linear problems, this value can be 1, unless more than 50 inner iterations are required, when maximum outer iterations could be as large as 10. For non linear problems, this values should be larger, but generally less than 100.

Maximum inner iterations

This value is the maximum number of iterations per call to the solver. The default value is 50, and this is sufficient for most problems.

Matrix Preconditioning Method

Use this radio group to choose the matrix conditioning method. The default is to use a modified incomplete cholesky method.

Relaxation Parameter

This value is used if the modified incomplete cholesky method is used. Usually, this value is 1.0, but for some problems a value of 0.99, 0.98, or 0.97 will reduce the number of iterations required for convergence.

Head Change Criterion For Convergence

When the maximum absolute value of head change from all cells during an iteration is less than or equal to the head change criterion, AND the residual change criterion is met, solver iteration stops.

Residual Change Criterion For Convergence

When the maximum absolute value of residuals from all cells during an iteration is less than or equal to the residual change criterion, AND the head change criterion is met, solver iteration stops.

Reset

This button will change all options to the default state.

RCH Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Recharge package is used to simulate recharge to an aquifer due to rainfall and infiltration. Recharge is typically defined by specifying a recharge value for each stress period for each vertical column in the grid (i.e., a NLAY X NCOL array of values is entered). The recharge value represents the amount of water that goes into the groundwater system and not the amount of precipitation. The units of recharge are length/time, so in a cell with a recharge value of 10(m/day), 10 meters of rainfall each day would be distributed across every m^2 of the cell. The recharge parameters are specified in the Recharge Package dialog. The options in the dialog are as follows:

Recharge Option

Three recharge options are supported by MODFLOW: recharge only at the top layer, recharge at specified vertical cells, and recharge at highest active cells.

Multiplier

The multiplier is a constant which can be written to the package file with each stress period array. Each value in the array is scaled by the multiplier as the array is imported to MODFLOW. The format button brings up a dialog listing the standard MODFLOW formats. This format is used for displaying the values in the spreadsheet and it controls how the values are written to the package file.

Stress Periods

The values displayed in the spreadsheet are for an individual stress period. The values associated with other stress periods can be edited by entering the ID of the desired stress period in the Stress period edit box in the center of the dialog. If the Use previous option is selected for a given stress period, the values from the previous stress period are used and the spreadsheet is dimmed.

Layer Indicator

If the Recharge at specified vertical cells option is chosen, the layer indicator for each vertical cell can be displayed and edited in the spreadsheet window by selecting Layer indicator in the View/Edit option.

Constant -> Array

The **Constant -> Array** button brings up a dialog which prompts for a single value. This constant is then assigned to each item in the array for the given stress period.

2D Data Set -> Array

The **2D Data Set -> Array** button brings up the Data Browser listing all of the current data sets associated with the current 2D grid. In order for this button to be active, the 2D grid must have the same number of rows and columns as the 3D grid. The selected data set is copied to the recharge array. Data sets are typically generated with the 2D Scatter Point module. The 2D Scatter Point module can be used to interpolate from a scattered set of rainfall measurements to the cell locations. If the data set is transient, the values in the data set are linearly interpolated, if necessary, to each stress period as the data set is copied to the array.

Array -> 2D Data Set

The **Array -> 2D Data Set** button copies the array to the 2D data set list associated with the existing 2D grid.

Spreadsheet

The recharge flux values for each of the vertical columns in the grid are displayed and edited in the spreadsheet at the lower part of the dialog.

Reset

The **Reset** button deletes all of the data currently defined in the package and restores the package parameters to the default values.

PEST ASP Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Prior to version 9.0 this command and dialog were called "Convergence Options".

When performing automated parameter estimation, a set of optimal parameter values can only be found if MODFLOW produces a stable solution for all iterations. One of the things that can cause problems with the inverse model is cells going dry. The dry cell flag that is written to the head array can cause instability in the inverse model, and thus it will not converge. When such problems occur, a successful run may be achieved by turning on some convergence options that have been added to the GMS version of MODFLOW by John Doherty. These options are accessed via the ***PEST ASP Package*** command in the *MODFLOW* menu. The options are as follows:

Do Not Stop on Non-Convergence

The "Do not stop ..." option is used for transient simulations. When this option is on, if the MODFLOW model does not converge for a particular time step, it continues to the next time step rather than aborting.

Assign Bottom Elevation to Dry Cells

During a normal MODFLOW run, if a cell goes dry, the HDRY value assigned in the BCF package is assigned to the cell. The default value for HDRY is -888. At each iteration of the inverse model run (each time a MODFLOW solution is found), the inverse model interpolates the computed head values to the observation points and computes the residual error. This error is factored into the objective function. If the head at a cell near an observation point suddenly switches from a normal head value to -888, an extreme discontinuity is introduced to the objective function and the inverse model may not be able to converge. This problem can be avoided by turning on the Assign bottom elevation to dry cells option. If this option is on, the bottom elevation associated with the cell is assigned as the head value for the cell.

Prevent Cell Drying

The Prevent Cell Drying option does not allow a cell in the bottom layer to go dry. The head is not allowed to drop below the value corresponding to the specified minimum saturated thickness. Additional instruction is provided by Richard Winston and in a USGS Open-File [1]:

> Use of the wetting capability can cause serious problems with convergence. You can try to avoid this by several methods.

1. If you know a cell should never become wet, make it an inactive cell rather than a variable head cell.
2. You can adjust the value of the wetting threshold in WETDRY. (Higher is more stable but may be less accurate.)
3. You can decide which neighbors will be checked to decide if a cell should be wetted using WETDRY. Often it is better to allow only the cell beneath the dry cell to rewet it.
4. You can use IHDWET to determine which equation is used to specify the head in newly wetted cells.
5. You can vary the wetting factor WETFCT.
6. In steady-state conditions you can adjust initial conditions to values that are close to your best guess of the final conditions to improve stability.
7. You can choose a different solver. The SIP, PCG1, and PCG2 solvers will work with the wetting capability. The SOR solver doesn't work well with the wetting capability. Note that cells can not change between wet and dry during the inner iterations of the PCG1 and PCG2 solvers. The PCG1 solver is no longer included in the USGS version of MODFLOW.
8. When using the PCG2 solver, you can set RELAX in the range of 0.97 to 0.99 to avoid zero divide and non-diagonally dominant matrix errors. (However, this is an infrequent cause of instability. If such an error occurs, PCG2 prints an error message in the output file and aborts the simulation.)
9. When using the PCG2 solver, you can set DAMP to a value between 0 and 1.
10. Unrealistically high conductances on boundary cells can contribute to instability. Check the conductances in the Drain, River, Reservoir, Lake, Stream, and General-Head Boundary packages. In the Evapotranspiration check the EVT Flux Stress[i] and EVT Extinction Depth which together control the conductance of evapotranspiration cells.

    The two most important variables that affect stability are the wetting threshold and which neighboring cells are checked to determine if a cell should be wetted. Both of these are controlled through WETDRY. It is often useful to look at the output file and identify cells that convert repeatedly from wet to dry. Try raising the wetting threshold for those cells. It may also be worthwhile looking at the boundary conditions associated with

dry cells.

Sometimes cells will go dry in a way that will completely block flow to a sink or from a source. After that happens, the results are unlikely to be correct. It's always a good idea to look at the flow pattern around cells that have gone dry to see whether the results are reasonable.

References

[1] http://pubs.er.usgs.gov/usgspubs/ofr/ofr91536

RIV Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

A set of selected cells can be specified as river cells using the ***Point Sources/Sinks*** command in the *MODFLOW* menu or river cells can be created using the conceptual model in the Map module. River attributes are typically used with arcs but may also be associated with polygons and points. The river parameters include elevation, stage, and conductance. Elevation is constant. The river stage and conductance may either be constant or vary with time.

When the river attribute is assigned to an arc, the conductance is applied uniformly over the arc, but separate elevations and stage values are applied to each of the nodes on the ends of the arc, and the elevation and stage are assumed to vary linearly between the nodes. When a river object is defined using a polygon or a point, all of the values are assigned directly to the cell(s) overlapped by the polygon or point.

Once a set of river cells has been specified, the *MODFLOW River Package* dialog can be used to view and edit the values assigned to the cells. The options in the dialog are as follows:

Stress Period

This field shows the current stress period and allows you to cycle through the stress periods and view the different river properties for those stress periods in the spreadsheet below.

Spreadsheet

For cells where river type boundary conditions have been assigned, the stage, conductance, bottom elevation, and conductance factor assigned to each cell are displayed in the spreadsheet portion at the lower part of the dialog. The spreadsheet can be used to edit the row, column, layer, stage, conductance, and bottom elevation values. For a transient simulation, the values displayed in the spreadsheet are for an individual stress period. The values associated with other stress periods can be edited by entering the number of the desired stress period in the stress period edit box in the center of the dialog. If the Use previous option is selected for a given stress period, the values from the previous stress period are used and the spreadsheet is dimmed. The conductance factor is used with RIV parameters. When doing parameter estimation the value of river conductance assigned to a particular river boundary condition will be the value of the river parameter multiplied by the conductance factor. For more information on RIV parameters and conductance factors see the MODFLOW 2000 manual.

SIP Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The SIP package is one of four solver packages for MODFLOW supported by GMS.

Maximum Iterations Per Time Step

This value is the maximum number of times through the iteration loop in one time step in an attempt solve the system of finite-difference equations.

Number of Iteration Parameters

Five variables are generally sufficient.

Acceleration Parameter

This value must be greater than zero and is usually 1.0.

Head Change Criterion For Convergence

When the maximum absolute value of head change from all cells during an iteration is less than or equal to the head change criterion, solver iteration stops.

Print Out Interval

The maximum absolute head change is printed for each iteration of a time step whenever the time step is an even multiple of the print out interval. The printout also occurs at the end of each stress period regardless of the print out interval.

Iteration Parameter Seed

These options control the seed for calculating iteration variables. You can either specify the seed yourself or have MODFLOW calculate it.

Reset

This button will change all options to the default state.

SSOR Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The SSOR package is one of four solver packages for MODFLOW supported by GMS. The SOR solver only uses a head change convergence criterion and may result in significant volume budget error.

Maximum iterations per time step

This value is the maximum solver iterations per timestep. The default value is 50.

Acceleration Parameter

This value is usually between 1.0 and 2.0

Head change criterion for convergance

When the maximum absolute value of head change from all cells during an interation is less than or equal to the head change criterion, solver iteration stops.

Print out interval

The maximum absolute head change is printed for each iteration of a time step whenever the time step is an even multiple of the print out interval. The printout also occurs at the end of each stress period regardless of the print out interval.

Reset

This button will change all options to the default state.

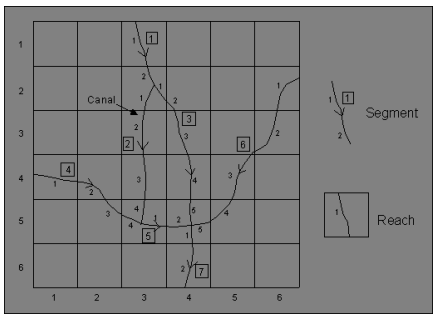
STR/SFR Packages

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The STR and SFR packages are used to simulate the interaction between surficial streams and the groundwater. These packages are somewhat similar to the River package in that water can move from the stream to the aquifer or from the aquifer to the stream depending on the relative differences in the stream stage and the water table elevations. However, unlike the River package, flow is routed through the stream. For the STR package, simple channel hydraulics and Manning's equation are used to compute the stage in the stream. The SFR package has more sophisticated hydraulics and routing options.

Stream Network

When either the STR or SFR package is used, a complete, ordered stream network must be defined as shown in the following figure.



Ordered stream network used by the STR or SFR packages.

A stream network is composed of reaches and segments. A reach is the portion of a stream that lies inside a single cell. A single cell may contain multiple reaches. A segment is a group of reaches that forms one section of the stream. The reaches within a segment are always numbered from upstream to downstream. The segments should also be numbered from upstream to downstream.

The incoming flowrate may only be defined for the topmost segments (segments 1, 4, and 6 in the above figure) and is typically entered as a positive value to represent the amount of flow in the stream channel. Flow is then routed and combined to get the incoming flowrate for the other segments. An exception to this is a diversion such as the canal at segment #2 in the above figure. In this case the flowrate into the diversion should be specified. The flow to the diversion would be subtracted from the flow coming out of segment #1 and whatever is left over would be routed to segment #3.

Defining Streams

Unlike other packages in GMS, a stream network cannot be created by selecting cells in the grid and manually assigning values to the cells. Due to the complex nature of the input and the requirement that the data be ordered in a specific fashion, a stream network can only be created using a series of Stream Arcs in the Map module. The cell-by-cell values are automatically created and properly ordered when the ***Map -> MODFLOW*** command is selected.

There are some restrictions on the STR and SFR conceptual models:

- STR and SFR stream arcs cannot both exist in the same coverage because of the stream numbering issues.
- All STR (or SFR) arcs must be in one coverage. You cannot use two or more coverages to represent your stream network.
- SFR - All downstream streams must be diversions except for one. The variable OUTSEG is the segment number of the downstream segment that is not a diversion.
- SFR - If the current segment has the diversion flag set then IUPSEG is the id of the upstream segment.

Set Arc Directions From Stream Data

Right-clicking on a coverage containing STR streams shows a *Set Arc Directions From Stream Data* menu item. This operation will cause all stream arcs to point downhill based on the elevation of the top of the river sediments as defined at the nodes.

STR Package

In the input file for the STR package, each reach is assigned the following values:

- ijk indices of cell
- segment ID
- reach ID
- stage (typically starting stage)
- conductance

- conductance factor (multipled by the conductance during parameter estimation)
- Elevation of the bottom of the streambed
- Elevation of the top of the streambed

Each segment is assigned the following values:

- incoming flow (for top segments and diversions)
- width
- slope
- roughness coefficient
- IDs of tributaries (upstream segments)
- ID of upstream segment (for diversions only)

When Manning's equation is used to route the flow through the network, the stream channels are assumed to have a rectangular cross section.

When the Map -> MODFLOW command is selected, GMS automatically does the following: classifies all of the cells beneath the streams as reaches, builds segments, numbers the reaches and segments, and assigns the appropriate values to the reaches and segments. These values defined at the nodes are linearly interpolated across the arcs when the model is converted. The slope assigned to reaches is computed by dividing the difference in the streambed top elevations at the ends of the arc by the arc length multiplied by the sinuosity factor.

The STR Package Dialog

Once the ***Map -> MODFLOW*** command is selected and the stream data have been assigned to the grid cells, the data can be viewed and some of the input values can be edited using the *Stream Package* dialog. The options for the Stream Package dialog include:

Calculate Stage

The Calculate stage toggle is used to specify whether the Stream package should compute the stages (specified stages are initial values only) or whether the specified stages should be used directly.

SFR Package

Support for the SFR2 package was added at GMS version 7.0.

The SFR Package Dialog

Once the ***Map -> MODFLOW*** command is selected and the stream data have been assigned to the grid cells, the data can be viewed and some of the input values can be edited using the SFR Package dialog. Stream segments and reaches cannot be created in the *SFR Package* dialog - creation can only be done via the ***Map -> MODFLOW*** command. The *SFR Package* dialog has two spreadsheets, one showing stream segments and one showing stream reaches. For a further explanation of the columns in each spreadsheet you can refer to the Online Guide to MODFLOW [30].

ISTCB2 info

If the *Save ISTCB2 info to *.istcb2 text file* option is on, MODFLOW will save more stream related data to a text file which has the same prefix as the other MODFLOW files and a "*.istcb2" extension. This text file will be displayed in the MODFLOW Solution in the Project Explorer when the solution is imported. The data in this file includes information on "inflows and outflows from each reach; on stream depth, width, and streambed conductance; and on head difference and gradient across the streambed."

If the *Save ISTCB2 info to *.istcb2 text file* option is off, streamflow out of each reach will be saved to the CCF file if the *Save cell by cell flow terms to *.ccf file* is on in the*Output Control* dialog.

Starting at GMS 9.0, this information is saved to a second CCF file that will appear in the solution as "CCF2". This is so that the stream flow out of each reach is not included in the flows in and out of the aquifer in the flow budget. The total stream flow out of each reach is displayed at the bottom of the flow budget.

UZF Package

The UZF Package is used to simulate percolation of water through the unsaturated zone. This package is compatible with MODFLOW 2005 (not earlier versions of MODFLOW).

The inputs to the UZF Package are edited in the UZF Package dialog. The inputs to the UZF package are explained here [1]. All of the inputs are labeled using the MODFLOW variable names.

Gages

The spreadsheet with the IUZROW, IUZCOL, IFTUNIT, and IUZOPT columns is used to edit gages associated with the UZF package. By default GMS will assign the unit numbers to the gage files (the IFTUNIT variable). However, the user can specify these values by turning on the toggle below the spreadsheet and entering the unit numbers in the IFTUNIT column. Gages can be added or removed by selecting the insert or delete buttons below this spreadsheet.

Care should be taken to enter unique unit numbers. The user can inspect the MODFLOW name file (*.mfn) written by GMS to ensure that the specified unit numbers are unique.

Multiplier

The multiplier is a constant which can be written to the package file with each array. Each value in the array is scaled by the multiplier as the array is imported to MODFLOW.

Stress Periods

The values displayed in the spreadsheet are for an individual stress period for the first 4 arrays (FINF, PET, EXTDP, and EXTWC). The other arrays are not specified per stress period.

The values associated with different stress periods can be edited by entering the ID of the desired stress period in the Stress period edit box. If the Use previous option is selected for a given stress period, the values from the previous stress period are used and the spreadsheet is not editable.

Constant -> Array

The **Constant -> Array** button brings up a dialog which prompts for a single value. This constant is then assigned to each item in the array for the given stress period.

2D Data Set -> Array

The **2D Data Set -> Array** button brings up the Data Browser listing all of the current data sets associated with the current 2D grid. In order for this button to be active, the 2D grid must have the same number of rows and columns as the 3D grid. The selected data set is copied to the recharge array. Data sets are typically generated with the 2D Scatter Point module. The 2D Scatter Point module can be used to interpolate from a scattered set of rainfall measurements to the cell locations. If the data set is transient, the values in the data set are linearly interpolated, if necessary, to each stress period as the data set is copied to the array.

Array -> 2D Data Set

The **Array -> 2D Data Set** button copies the array to the 2D data set list associated with the existing 2D grid.

Spreadsheet

The values for the array listed in the View/Edit selection box are shown in the spreadsheet.

References

[1] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/uzf_unsaturated_zone_flow_pack.htm

WEL Package

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

A set of selected cells can be specified as wells using the ***Point Sources/Sinks*** command in the *MODFLOW* menu or by using the conceptual model in the Map module. Wells are specified by assigning a pumping rate to a selected cell at the location of each well. Wells can be either injection wells (positive flow rate) or extraction (negative flow rate) wells.

Once a set of cells has been specified, the *Well Package* dialog can be used to view and edit the values assigned to the cells.

Stress Period

The stress period field shows the current stress period and allows you to cycle through the stress periods and view the different well properties for those stress periods in the spreadsheet below.

Use Previous

Use this option on a stress period to use the well values from the previous stress period for the current stress period. This option is unavailable for steady state models and the first stress period of transient models.

Spreadsheet

For cells where well type boundary conditions have been assigned, the well flow rate for each cell is displayed in the spreadsheet portion at the lower part of the dialog. The spreadsheet can be used to edit the well name, row, column, layer, and flow values.

For a transient simulation, the values displayed in the spreadsheet are for an individual stress period. The values associated with other stress periods can be edited by entering the number of the desired stress period in the stress period edit box in the center of the dialog. If the Use previous option is selected for a given stress period, the values from the previous stress period are used and the spreadsheet is dimmed.

The conductance factor is used with WEL parameters. When doing parameter estimation the value of well conductance assigned to a particular well boundary condition will be the value of the well parameter multiplied by the conductance factor. For more information on WEL parameters and conductance factors see the MODFLOW 2000 manual.

Display Cell IDs/IJK

The radio group at the bottom of the dialog allows the user to adjuster whether the cell ID or the cell i, j, k values are displayed for the cells containing the boundary condition/source/sink.

Delete All

The Delete All button deletes all of the data currently defined in the River package and restores the River package parameters to the default values.

6.3.2. MODFLOW Pre-Processing

Building a MODFLOW Model

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Two basic approaches are provided in GMS for constructing a MODFLOW model: the model can be completely defined using the tools in the 3D Grid module, or the model can be defined with the aid of the feature objects in the Map module. Also, scatter points and solids can be used to define layer elevation, and boreholes can be used with TPROGS to create material sets for the 3D grid.

Basic Steps in Building a MODFLOW Model

The basic steps in building a MODFLOW model include:

- Build a 3D grid defining the extents of the model. Two approaches are available:
  - MODFLOW Grid Approach
  - MODFLOW Conceptual Model Approach
- Initialize MODFLOW
- Assign boundary conditions and model stresses, including sources and sinks
- Define layer elevations
- Assign material properties
- Run MODFLOW

MODFLOW Array Editor

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The MODFLOW Array Editor is used throughout the MODFLOW interface to edit array based data. It includes the following features:

Spreadsheet

The array is displayed in a spreadsheet and each value can be selected and edited. Individual layers in the array are displayed one at a time. The edit fields in the spreadsheet associated with inactive cells are dimmed.

Layers

The edit field in the upper left corner of the dialog is used to switch to a different layer.

Multiplier

The multiplier is a value that can be specified to scale all of the values in the array.

Constant -> Grid

The **Constant -> Grid** button prompts for a single value and assigns the value to all of the cells in the entire array.

Constant -> Layer

The **Constant -> Layer** button prompts for a constant value which is assigned to all of the cells in the currently displayed layer.

3D Data Set -> Grid

The **3D Data Set -> Grid** button brings up the Select Data Set dialog listing all of the current data sets associated with the 3D grid. The selected data set is then copied into the starting heads array. This option is typically used to load in a previously computed MODFLOW solution to use as the initial condition. The previously computed solution must be imported as a data set before selecting this option.

2D Data Set -> Layer

The **2D Data Set -> Layer** button allows the user to select one of the data sets associated with a 2D grid and copy it to the current layer of the starting heads array. In order for this button to be active, a 2D grid that has the same number of rows and columns as the 3D grid must be imported or created. Such a grid can be automatically generated using the ***Grid -> 2D Grid*** command in the *Grid* menu. The 2D Data Set -> Layer option is typically used to load in a data set that has been created by interpolating heads from a 2D scatter point set to the 2D Grid.

Grid -> 3D Data Set

The Grid -> 3D Data Set button copies values from the entire starting heads array to the 3D grid data set list.

Layer -> 2D Data Set

The Layer -> 2D Data Set button copies values from the selected layer of the starting heads array to the 2D grid data set list.

MODFLOW Grid Approach

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Building a MODFLOW 3D Grid

For models with simple geometry and boundary conditions, the entire model can be constructed using the tools and commands in the 3D Grid module. With this approach, the editing of the MODFLOW data is performed directly on the grid on a cell-by-cell basis. The main steps are as follows:

- Create a 3D cell-centered grid covering the domain to be modeled using the ***Create Grid*** command in the *Grid* menu.
- Use the commands in the MODFLOW menu to initialize and define the data required by the MODFLOW packages.
- Boundary conditions such as wells are defined by selecting the cells and assigning the attributes directly to the cells.

Defining the Layer Data

An important part of a MODFLOW model is the definition of the layer data (hydraulic conductivity, layer elevations, leakance, etc.). While both the Grid and Map module approaches to constructing a MODFLOW model can be used to define the layer data, both approaches may lead to an overly simplistic definition of the stratigraphy. Layers with spatially varying thicknesses can be handled most effectively using a special set of layer data tools provided in GMS. Solids can also be used to define layer data with the ***Solids->MODFLOW*** and ***Solids->HUF*** commands.

MODFLOW Conceptual Model Approach

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

A MODFLOW model can be created in GMS using one of two methods: assigning and editing values directly to the cells of a grid (the grid approach), or by constructing a high level representation of the model using feature objects in the Map module and allowing GMS to automatically assign the values to the cells (the conceptual model approach). Except for simple problems, the conceptual model approach is typically the most effective.

Definition

In GMS, the term conceptual model is used in two different ways. In the generic sense, a conceptual model is a simplified representation of the site to be modeled including the model domain, boundary conditions, and material zones. GMS also has a conceptual model object, that can be defined in the Map module. The conceptual model object can contain coverages.

A conceptual model object can be defined in the Map module using feature objects, such as points, arcs, and polygons. The conceptual model is constructed *independently* of the numerical grid. Once the conceptual model object is defined, a grid can be automatically generated to fit the conceptual model. The boundary conditions and model parameters are computed and assigned to the proper cells. The MODFLOW data are converted from the conceptual model to the cells of the grid. This approach to modeling fully automates the majority of the data entry

and eliminates the need for most or all of the tedious cell-by-cell editing traditionally associated with MODFLOW modeling. Once the conceptual model is constructed and converted to a grid, the package dialogs and interactive editing tools in the *MODFLOW* menu can be used to edit or review the data if desired.

A complete conceptual model object consists of several coverages. One coverage is typically used to define the sources and sinks such as wells, rivers, lakes, and drains. Another coverage (or the same coverage) is used to define the recharge zones. Other coverages can be used to define the zones of hydraulic conductivity within each layer. Any number of coverages may be used, or all these attributes may exist in the same coverage.

In addition to the feature data, a conceptual model may include other data (scatter points, boreholes, solids) to define the layer elevations. A specialized set of tools for manipulating layer elevation data is provided in GMS.

Steps in Developing a Conceptual Model

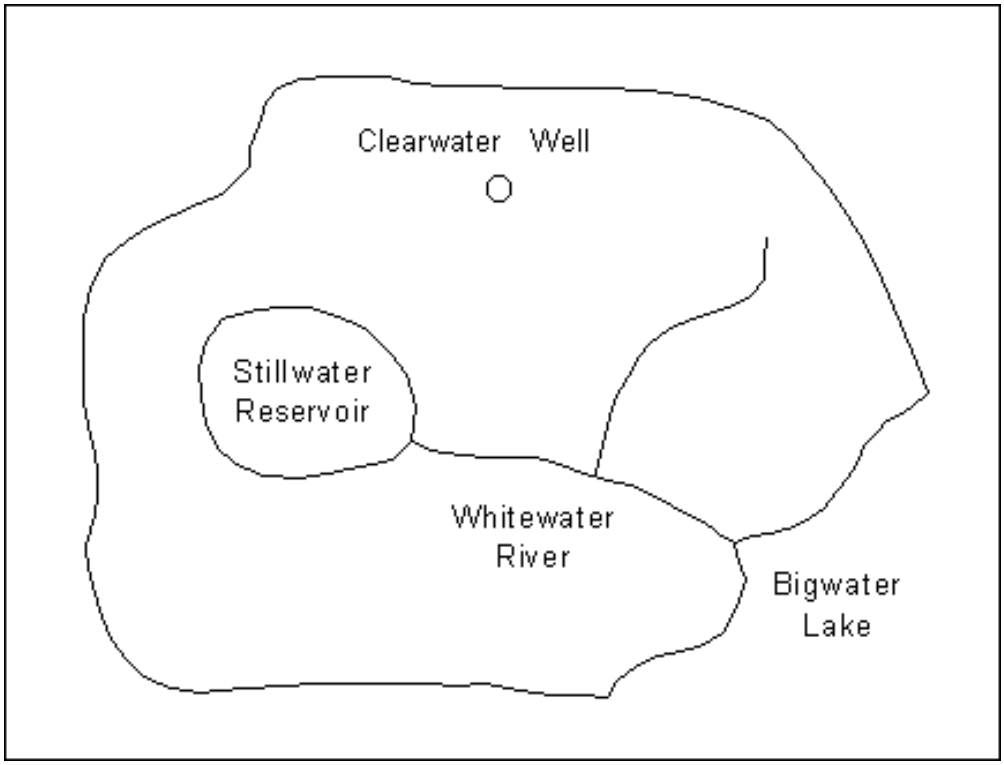
Several steps are involved in setting up a MODFLOW conceptual model and converting the conceptual model to a numerical model. The basic steps are as follows:

1. Create a MODFLOW conceptual model in the Map Module.
2. Decide how many coverages you will need and what attributes you want with each. It is typical to use three coverages, each with a different purpose. One coverage is used for sources and sinks, like wells, specified head boundaries etc. Another coverage is used for recharge, and another is used for hydraulic conductivity. However, it is not required to use coverages in this manner, and the attributes associated with a coverage are user defined, so you can define a coverage with any combination of attributes. Thus, if your recharge zones were the same as your hydraulic conductivity zones, you might use just one coverage for both.
3. Create the sources and sinks. The most effective way to do this is with the aid of a background Image. A digital image representing a scanned map or an aerial photo of the site can be imported and displayed in the background using the image tools. Once the image is displayed, feature objects defining the model boundary, rivers, lakes, flow barriers, and specified head boundaries can be created on top of the background image.
4. Create the areal features, such as recharge zones and evapotranspiration zones.
5. Create the layer features, such as hydraulic conductivity zones.
6. Use the ***Grid Frame*** command to place an outline of the numerical grid on the conceptual model. The frame is placed so that it just surrounds the conceptual model. The frame can be rotated if necessary if the major axis of the model is at an angle.
7. Use the ***Feature Objects | Map -> 3D Grid*** command to automatically generate a grid. The location of the grid is controlled by the Grid Frame and the density of the grid is automatically adjusted around user-specified points (typically wells).
8. Define the active region of the grid using the Activate Cells in Coverage command. This automatically activates all of the cells within the boundary of the conceptual model and inactivates all cells outside the boundary.
9. Initialize the MODFLOW data by selecting the ***New Simulation*** command in the *MODFLOW* menu. Select the type of mode (steady state vs. transient) and define stress periods if necessary. Define a set of starting heads. Go to the LPF/BCF/HUF Package dialog and define the layer type for each of the layers in the grid.
10. Select the ***Map -> MODFLOW*** command to automatically assign the MODFLOW boundary conditions, stresses, and material properties to the appropriate cells in the grid.
11. Use the specialized interpolation tools to define the layer elevations.

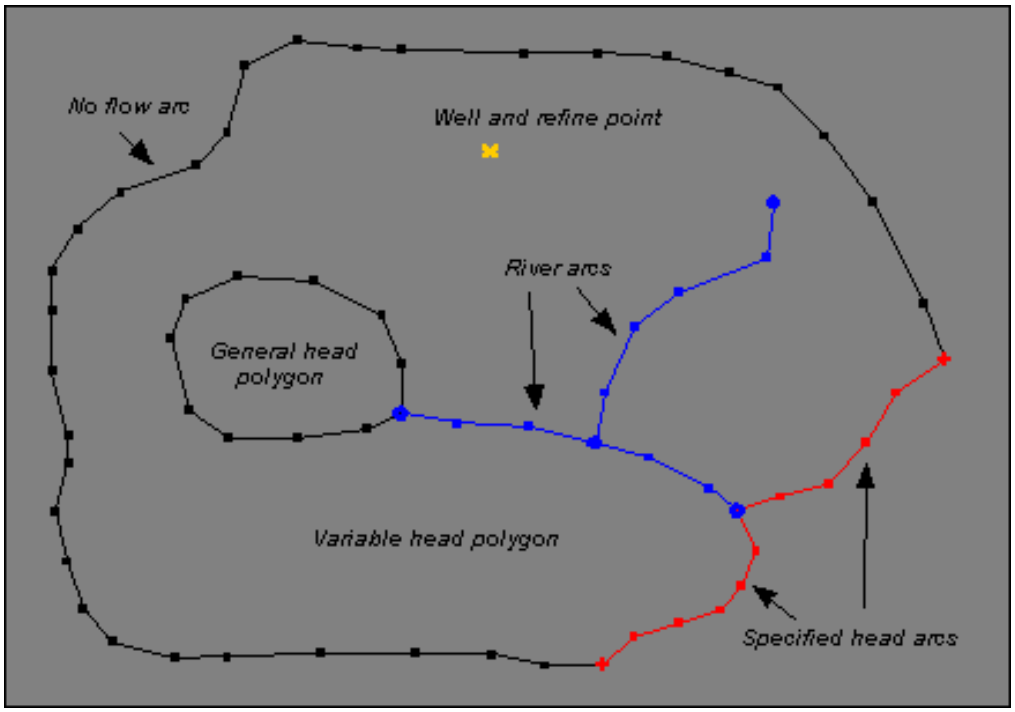
In many cases, it is useful to repeat some, but not all, of these steps. For example, suppose after running a simulation it is determined that one of the boundaries of the model corresponding to a groundwater divide is not properly located. The boundary can be moved by simply selecting and dragging the vertices and nodes of the arc(s) defining the boundary. Once the boundary is moved, step 6 should be repeated to redefine the active/inactive regions and then step 8 is repeated to reassign the model data to the cells.

Example Application

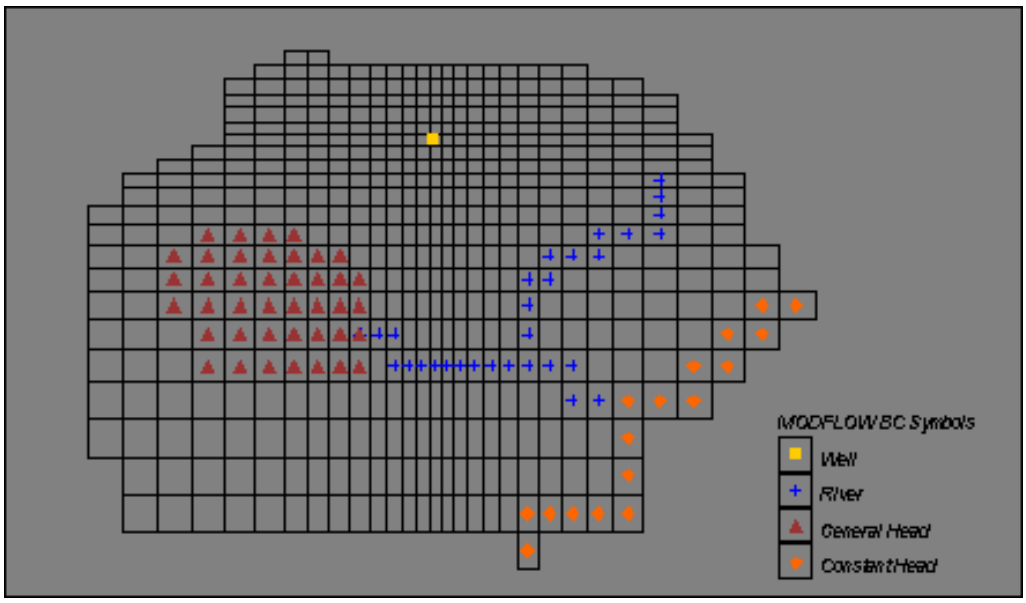
To illustrate the conceptual model approach, consider the site shown in the following figure. It represents information that might be available from a combination of sources including maps, photos, and GIS data. This information may include the location of hydrologic features as well as hydrologic properties of the site. However, this information is not yet organized into a form that is useful to a numerical model.



The first step in creating a conceptual model of this site is to create points, arcs and polygons that represent hydrologic features at the site. These points, arcs and polygons are assigned types that correspond to the feature they represent. Based on the attribute type, parameters such as head, concentration and conductance are assigned to these feature objects. The resulting coverage is shown in the following figure. Other coverages, defining such things as recharge zones, would also be defined.



The final step is to take the information that is stored in the conceptual model and construct a numerical model. GMS automates both the creation of the grid geometry as well as assigning boundary conditions and material parameters to the grid. Refine points can be used to specify areas where the grid should have a high density. By specifying polygons that represent the domain of the model, it is possible to automatically inactivate all the cells that lie outside that domain. Boundary conditions may be applied to the individual cells that are intersected by specified feature objects. In addition to determining which cells are assigned boundary conditions, GMS also calculates the appropriate values to assign to each stress period of a transient simulation. The resulting numerical model is illustrated in the following figure.



Advantages of the Conceptual Model Approach

There are numerous benefits to the conceptual model approach. First of all, the model can be defined independently of the grid resolution. The modeler does not need to waste valuable time computing the appropriate conductance to assign to a river cell based on the length of the river reach within the cell. This type of computation is performed automatically. Furthermore, transient parameters such as pumping rates for wells can also be assigned independently of model discretization. Transient parameters are entered as a curve of the stress vs. time. When the conceptual model is converted to the numerical model, the transient values of the stresses are automatically assigned to the appropriate stress periods. Since the conceptual model is defined independently of the spatial and temporal discretization of the numerical model, the conceptual model can be quickly and easily changed and a new numerical model can be generated in seconds. This allows the modeler to evaluate numerous alternative conceptual models in the space of time normally required to evaluate one, resulting in a more accurate and efficient modeling process.

A further advantage of storing attributes with feature objects is that the method of applying the boundary conditions to the grid cells reduces some of the instability that is inherent in finite difference models such as MODFLOW and MT3DMS. When the user enters individual values for heads and elevations, entering cell values one cell at a time can be tedious. It is also difficult to determine the correct elevation along a river segment at each cell that it crosses. The temptation is to select small groups of cells in series and apply the same values to all of the cells in the group. This results in an extreme stair-step condition that can slow or even prevent convergence of the numerical solver. By using GMS to interpolate values at locations along a linear boundary condition such as a river, the user insures that there will be no abrupt changes from cell to cell-thus minimizing the stair-step effect. It also produces a model with boundary conditions that more accurately represent real world conditions.

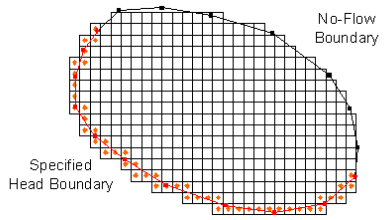
Defining the Layer Data

An important part of a MODFLOW model is the definition of the layer data (hydraulic conductivity, layer elevations, leakance, etc.). While both the Grid and Map module approaches to constructing a MODFLOW model can be used to define the layer data, both approaches may lead to an overly simplistic definition of the stratigraphy. Layers with spatially varying thicknesses can be handled most effectively using a special set of layer data tools provided in GMS. Solids can also be used to define layer data with the ***Solids->MODFLOW*** and ***Solids->HUF*** commands.

Activate Cells in Coverage

Once a grid has been created, the next step in converting a MODFLOW conceptual model to a numerical model is to inactivate the cells which lie outside the boundary of the conceptual model. This is accomplished by selecting the ***Activate Cells in Coverage*** command in the *Feature Objects* menu. When this command is selected, GMS utilizes the polygons in the coverage that is designated as the model boundary to determine which cells should be active and which cells should be inactive. Each cell is compared with the polygons in the coverage and if the cell does not lie within the interior of any of the polygons, the cell is determined to be outside the domain of the model and is inactivated. Cells inside the model domain are made active.

When cells are tested to determine whether they are outside or inside the model domain, if the cell lies partially inside the model domain and partially out, the attribute type of the feature object on the boundary where the cell is located is used to determine the active/inactive status of the cell. An example of this process is shown in the figure at the right.



Sample Application of the Activate Cells in Coverage Command

If an arc on the boundary of the model domain has no attribute assigned to it, it is assumed to be a no-flow boundary. If a cell is partially covered by a no-flow boundary, the cell is activated if the majority of the cell area is inside the coverage. Conversely, the cell is inactivated if the majority of the cell area is outside the coverage. As a result, the outer edges of the cells along the no-flow boundary approximately coincide with the no-flow arc.

If an arc on the boundary of the model domain has a head dependent attribute assigned to it, a different test is used. Any cell that intersects the arc is designated as active, regardless of what percentage of the cell is inside the model domain. As a result, the centers of the cells along the boundary approximately coincide with the source/sink arc. This is appropriate in this case since the stresses are applied in MODFLOW at the cell centers.

Stress Periods

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The computational time intervals for a MODFLOW simulation are called "stress periods". The transient stresses (pumping rates, river stages, etc.) can only change at the beginning of each stress period. Stress periods can be subdivided into smaller time steps, if desired. The Stress Periods button in the *Global Options\Basic Package* dialog is used to bring up the *Stress Period* dialog. The stress periods should be defined before any of the sources/sinks are entered. If the steady state option in the *Global Options\Basic Package* dialog is selected, the Stress Periods button is dimmed.

Each stress period is now a single row in the spreadsheet. The stress period properties such as length and number of time steps can now be copied quickly and easily to other stress periods using the spreadsheet interface.

Initialize

A set of stress periods can be defined using the Initialize button. The Initialize button brings up a dialog which is used to generate a set of stress periods of constant length. The dialog prompts for a number of stress periods, a length, a number of time steps, and a time step multiplier. A set of stress periods is then generated and displayed graphically in the stress period plot.

Stress Period Plot

The horizontal strip at the top of the Stress Periods dialog is used to graphically edit stress periods. A stress period can be selected in the plot by clicking on the stress period with the mouse. The values associated with the selected stress period can be edited in the spreadsheet.

Insert Buttons

New stress periods can be added by selecting a stress period and selecting either the Insert Row button or by editing the last row in the spreadsheet. Stress periods can be deleted by selecting the stress periods and selecting the Remove Row button .

Reference Time

The first cell in the spreadsheet is used to enter the reference time for the MODFLOW simulation. The reference time is the date/time corresponding to the beginning of the simulation (t=0). If the Use Date/Times option is selected in the Time display section, all time values entered for transient input data, i.e., time series defined in the XY Series Editor can be entered in a date/time format rather than a scalar time format. Also, when post-processing, the values shown in the time step selector in the Data Browser or at the top of the GMS Window are displayed in the date/time format. Furthermore, any time series curves entered as part of the MODFLOW conceptual model in the Map module that were defined using the date/time format will be automatically converted to the proper time scale when the conceptual model is converted to grid-based numerical model.

Map to MODFLOW

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

After the conceptual model is constructed and a grid has been created, the final step in converting a conceptual model to a MODFLOW numerical model is to select the Map -> MODFLOW command. However, before this command can be selected, MODFLOW must be initialized. MODFLOW is initialized as follows:

1. Switch to the 3D Grid module
2. Select the ***New Simulation*** command in the *MODFLOW* menu.
3. MODFLOW simulations are steady state by default. For a transient simulation, go to the *Global Options/Basic Package* Dialog and select the **Transient** option. Then set up the stress periods you wish to use in the simulation.
4. By default, the top layer is unconfined and the remaining layers are confined. To use a different set of layer types, go to the BCF/LPF/HUF Package dialog and select the appropriate layer type for each layer.

Once MODFLOW is initialized, the ***Map -> MODFLOW*** command becomes available. When the command is selected, the *Map -> MODFLOW Options* dialog appears. Three options are available for converting the conceptual model: Active coverage only, All applicable coverages, and All visible coverages. If the All applicable coverages option is chosen, all of the feature objects in all of the MODFLOW-related coverages in the active conceptual model are used. This option is typically selected when the conceptual model is first converted. If the Active coverage only or the All visible coverages option is selected, only a subset of the coverages are used to update the numerical model.

Temporal Discretization

See the Temporal Discretization page.

Multiple Values Per Cell

Because GMS processes each feature object separately, there will often be sources/sinks that were derived from two separate feature objects in the same cell. In fact, this is almost always the case in the cell that contains the endpoint of one arc and the beginning point of an adjacent arc. This is not an error. MODFLOW handles each of the boundary conditions in the cell simultaneously.

Specified Head Cells

Because the constant head condition forces the head in those cells to match whatever is specified, it is inappropriate to have other boundary conditions defined in the cells that are designated constant head. Therefore, GMS processes all of the specified head objects first. Afterwards, if there is another stress that should normally be assigned to a cell that has been previously assigned a constant head condition, the new stress is not assigned.

Changing Head Boundary

When mapping a specified head boundary to MODFLOW, GMS always uses the CHD package (Time Variant Specified Head package). In MF2K, a changing head boundary must be used in order to extract fluxes out of the MODFLOW output for the arcs in the map module. With MF2K you can not get a flux observation with normal Spec Head. It is assumed that most of the time the user would want to see the flux in/out of the boundary. If a cell contains two different arcs, the specified head is split in 2 pieces at cells where 2 specified head arcs meet. This is done for 2 reasons. First, the CHD package will combine the CHD boundary conditions that are in the same cell. Second, the flux in/out of the cell will be partitioned to the 2 different arcs in the map module.

A traditional specified head boundary can be manually applied by using the IBOUND and starting heads arrays. The computed heads from MODFLOW will be the same whether a constant head or changing head boundary is used.

Well Screens

NOTE: This approach was developed prior to the existance of the MNW package. The MNW package can more accurately represent a well that crosses multiple MODFLOW layers.

When using well screens in your conceptual model the following equation is used to partition the flow to different layers:

$$Q_i = \frac{T_i}{T} = \frac{(k_h)_i B_i}{\sum_{j=1}^{n} (k_h)_i B_i}$$

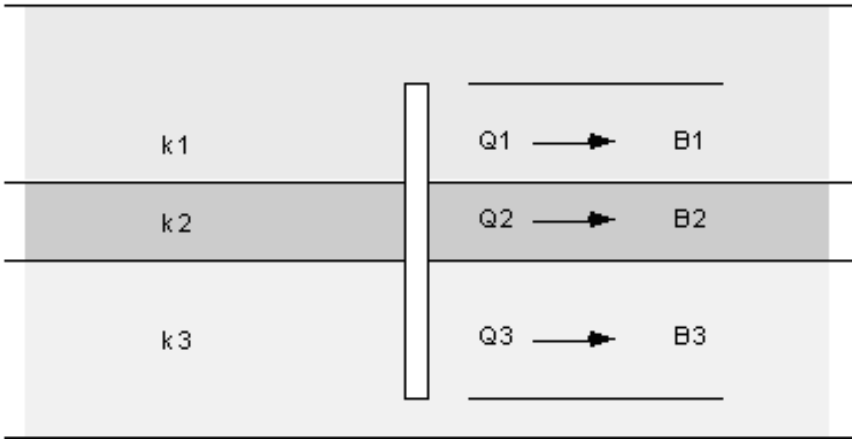
where

Q_i = The flow rate for layer i

$(kh)_i$ = The horizontal hydraulic conductivity for layer i

B_i = The length of the well screen intercepted by the layer

For example



Assuming the K's are the horizontal K's, Q_1 would be computed as:

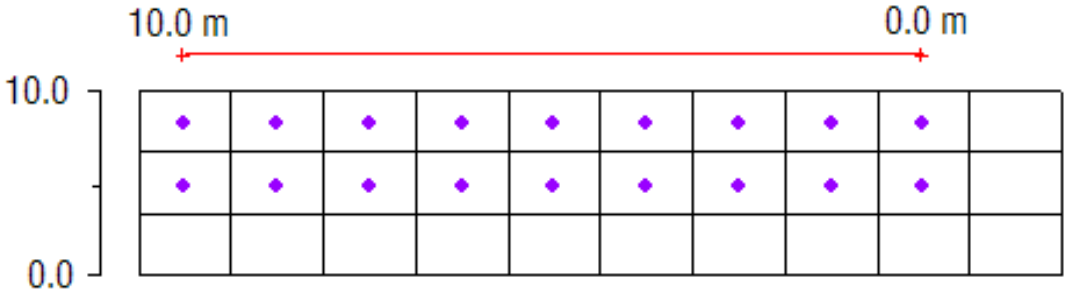
$$Q_1 = \frac{K_1 B_1}{K_1 B_1 + K_2 B_2 + K_3 B_3}$$

In order to complete this computation, the K's must be assigned to the cells in the grid. You must assign the K values to the grid prior to executing the ***Map -> MODFLOW*** command.

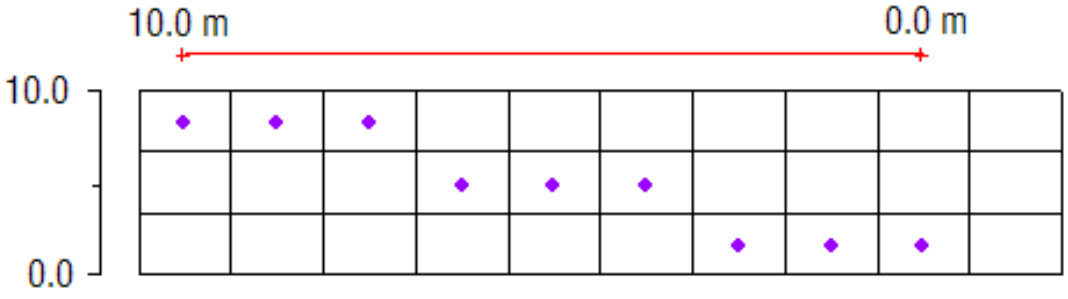
If the HUF package is being used then the K value for each cell is estimated from the HUF units.

Automatic Layer Assignment

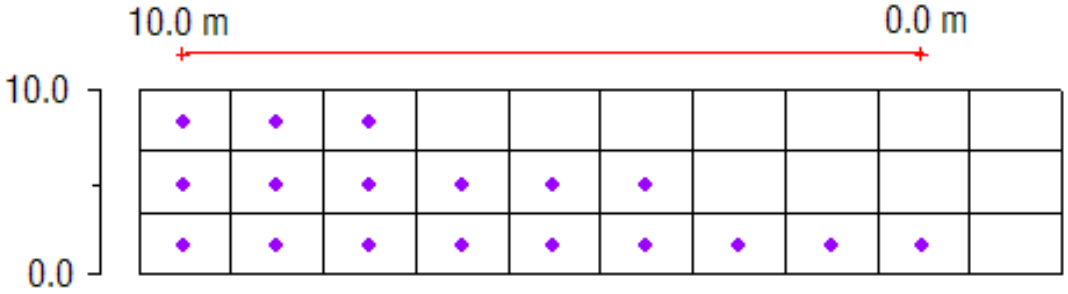
When building a MODFLOW conceptual model with a multi-layer model, it is necessary to define the range of layers associated with a particular source sink object. For example, an arc corresponding to a specified head boundary condition may be associated with layers 1-3 on the edge of a model. On the other hand, a drain arc in the middle of the model may only be associated with the top layer. The following example shows how a specified head boundary condition would map with the "Use layer range" option with the range set from 1 to 2:



In some cases, however, the proper layer for a particular source/sink object will depend on the elevation of the object relative to the layer elevations. For example, a drain may represent a channel that cuts through the terrain. In some places the channel may be shallow and correspond to layer 1. In other places, the channel may be deep enough that it cuts into layer 2. The "Auto assign to one cell" option is used for defining the layer range in these cases. If this option is selected, the object will be associated with the layer where the elevation or head associated with the object falls between the top and bottom elevation for the layer. The following shows how the same specified head boundary condition would map with the "Auto assign to one cell" option:



The "Auto-assign including lower cells" option is useful for cases where you want a specified head boundary assigned to the layer where the head is between the top and bottom elevation of the cell as well as any active cells below that cell. The following shows how the same specified head boundary condition would map with the "Auto-assign including lower cells" options:



Using TINs

TIN surfaces can be used as part of the conceptual model. See the Map to MODFLOW with TINs page.

Defining the Layer Elevations

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

One of the most important steps in defining a MODFLOW model is to define the layer elevations. The layer elevations are stored in the *Global Options/Basic package*. The layer elevations include a top elevation for layer 1 and a bottom elevation for all layers. For all layers except for the top layers, the top elevation for the layer is assumed to be equal to the bottom elevation of the layer above.

Define Layer Data

As described above, MODFLOW models can be defined using one of two approaches: (1) by editing the input values on a cell-by-cell basis directly on the grid, or (2) by creating a high level conceptual model using the Map module. Both approaches can be used to define the layer elevation arrays. With the grid approach, a constant value can be assigned to the entire array at once or to a set of selected cells. With the conceptual model approach, a set of polygons can be used to define zones of elevations within each layer. While these two approaches are simple to use, in most cases they result in an overly simplistic stair step definition of the layer elevations. In most cases, it is more appropriate to use the 2D geostatistical tools in GMS to smoothly interpolate layer elevations.

Interpolating Elevation Data

The following steps are used to interpolate elevation data:

Step 1 - Importing the Scatter Point Elevation Data

The fist step in defining MODFLOW layer elevation array data is to create a 2D scatter point set. The set should include a data set for the top elevation of the top layer and the bottom elevation array of the top layer and each of the underlying layers. A water table elevation corresponding to the desired initial condition (starting head) may also be defined. The simplest way to create such a scatter point set is to create a tabular scatter point file using a spreadsheet or a text editor. The scatter point file is imported ***File|Open*** menu command. A sample tabular scatter point file for a three layer model is shown below:

x	y	top1	bot1	bot2	bot3
360	1670	450	345	200	100
290	870	445	340	195	95
480	420	450	350	200	100
620	2120	455	245	200	100
990	1820	470	355	210	115
890	1190	465	350	205	110
1030	710	475	360	215	130
etc.					

Step 2 - Interpolating the Elevations to the MODFLOW Arrays

Once the scatter point file is imported to GMS, the next step in defining MODFLOW layer elevation arrays is to interpolate the elevations to the MODFLOW layer elevation arrays. The elevation values can be interpolated directly to the MODFLOW arrays using the ***...to MODFLOW Layers*** command in the *Interpolation* menu in the 2D Scatter Point module. When this command is selected, the Interpolate to MODFLOW Layers dialog appears.

The purpose of this dialog is to associate each of the data sets in the scatter point set with one of the layer data input arrays. A data set and the corresponding layer data array are selected in the top of the dialog and the Map button is selected. The defined relationship is then shown in the bottom of the dialog. Once this is completed for each data set/layer data array combination, the **OK** button is selected and the scatter point data set values are interpolated directly to the MODFLOW arrays using the currently selected interpolation options.

In some cases, GMS can automatically match the scatter point data sets to the appropriate layer elevation arrays. GMS searches each data set name to see if "top" or "bot" makes up any portion of the name. If so, it then searches for a number to determine the layer the array should be interpolated to. For example, the data set names top1, top of layer 1, and top elevation of layer 1 would all automatically map correctly.

It should be noted that the layer data can be set up using multiple scatter point sets. For example, it is possible to have one dense set of scatter points to define the ground surface (top of layer one), and a second, more sparse set of scatter points to define the layer bottom elevations. In this case, the to Layers command would need to be selected twice, once for each of the scatter point sets.

Step 3 - Fixing Layer Interpolation Errors

When interpolating layer data for the purpose of defining MODFLOW layer elevation arrays, there are often cases where the interpolated values overlap. For example, for some of the cells, the top elevation values for a particular layer may be lower than the bottom values for the layer. In some cases, the best way to fix such a problem is to experiment with the interpolation options or to create some "pseudo-points" to fill in the gaps between sparse scatter points. In other cases, the overlap may correspond to a pinchout or truncation in the layer. In such cases, the elevations need to be adjusted so that there is a small but finite thickness for all cells in the overlapping region.

The first step in fixing layer errors is to use the Model Checker to determine if elevation overlaps occur. If they do occur, the Fix Layer Errors button at the top of the Model Checker dialog can be used to bring up the *Fix Layer Errors* dialog. The number of overlap errors for each layer is listed on the right side of the dialog. A layer is highlighted and a correction method is selected on the left side of the dialog. The **Fix Selected Layer** button is then used to adjust the elevations. Four options are available for fixing layer errors:

Average

With the Average method, for each cell where an overlap is found, the average elevation at the overlap is computed as

$$\text{ave} = \frac{\text{top} - \text{bottom}}{2}$$

The top and bottom elevations are then adjusted as follows:

$$\text{top} = \text{ave} + \frac{\text{min thickness}}{2}$$

$$\text{bot} = \text{ave} - \frac{\text{min thickness}}{2}$$

This option is useful for modeling the transition zones adjacent to embedded seams.

Preserve Top

With the Preserve top method, at each cell, where an overlap is found the top elevation is unchanged and the bottom elevation is adjusted to:

$$\text{bot} = \text{top} - \text{min thickness}$$

This option can be used to model truncated outcroppings.

Preserve Bottom

With the Preserve bottom method, at each cell where an overlap is found the bottom elevation is unchanged and the top elevation is adjusted to:

$$\text{top} = \text{bot} + \text{min thickness}$$

Truncate to Bedrock

The Truncate to bedrock option differs from the other methods in that it can be used to alter several layers at once. With this method, it is assumed that the bottom elevation values for the bottom layer represent the top of a bedrock unit. The bedrock elevations may overlap several upper elevation arrays. Each cell in the grid is checked and if the bedrock elevation is above the top elevation for the cell, the cell is turned off (made inactive). If the bedrock elevation is below the top elevation and above the bottom elevation for the cell, the bottom elevation for the cell is set equal to the bedrock elevation. If the bedrock elevation is below the bottom of the cell, the cell elevations are unchanged.

Define Layer Data with Solids

Solids can also be used to define layer data for MODFLOW. The ***Solids -> MODFLOW*** and ***Solids -> HUF*** commands in the Solid module set up the layer elevations using the currently defined solids.

Standard MODFLOW Parameters

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

Caution

There are two ways to do MODFLOW parameterization in GMS:

1. The key value method **(recommended)**
2. The standard MODFLOW method

The "key value" approach is the preferred approach to defining parameters in GMS. Using key values is the only option for WEL, RIV, DRT, DRN, GHB, CHD, STR, and HFB parameters. In GMS, using the key value approach with these parameters provides the same functionality of parameter instances available in MODFLOW.

Key values can also be used with array based parameters. However, GMS does support defining array based parameters with clusters and instances (for ETS, EVT, RCH parameters).

When reading in a MODFLOW simulation that was created outside of GMS all parameters will be converted to key values so long as the parameter can be represented by key values.

Key values can not be used to represent an array based parameter if the parameter is defined using more than one cluster or more than one instance. Also, if more than one parameter uses the same zone array with a matching IZ value then the parameter can not be represented using key values. Further, if the multiplier array associated with the parameter has values other than 1.0 in the zones where the parameter is used then the parameter can not be represented with key values.

The standard MODFLOW method is considered an "advanced" feature in GMS. **Only users that understand how MODFLOW uses parameters should attempt to use this feature.**

This article describes how to use the standard MODFLOW parameterization method in GMS.

Support in GMS

Reading

GMS can read MODFLOW models that use standard MODFLOW parameters.

Writing

GMS writes MODFLOW models using the key value approach. So, although GMS can read models that use standard MODFLOW parameters, it cannot write them using the standard MODFLOW parameter method. Writing is usually done using the key value method. There are a few exceptions which include recharge and evapotranspiration parameter instances, and LPF and HUF parameter clusters. GMS will write these parameters using the standard MODFLOW parameter method because these cannot always be represented using the key value method.

Editing

When GMS reads a model that uses standard MODFLOW parameters it converts them to regular GMS key value approach parameters. The parameters can then be edited using the *Parameters* Dialog. Recharge, evapotranspiration, LPF, and HUF parameters are special in that you can edit the instance and cluster information associated with these types of parameters.

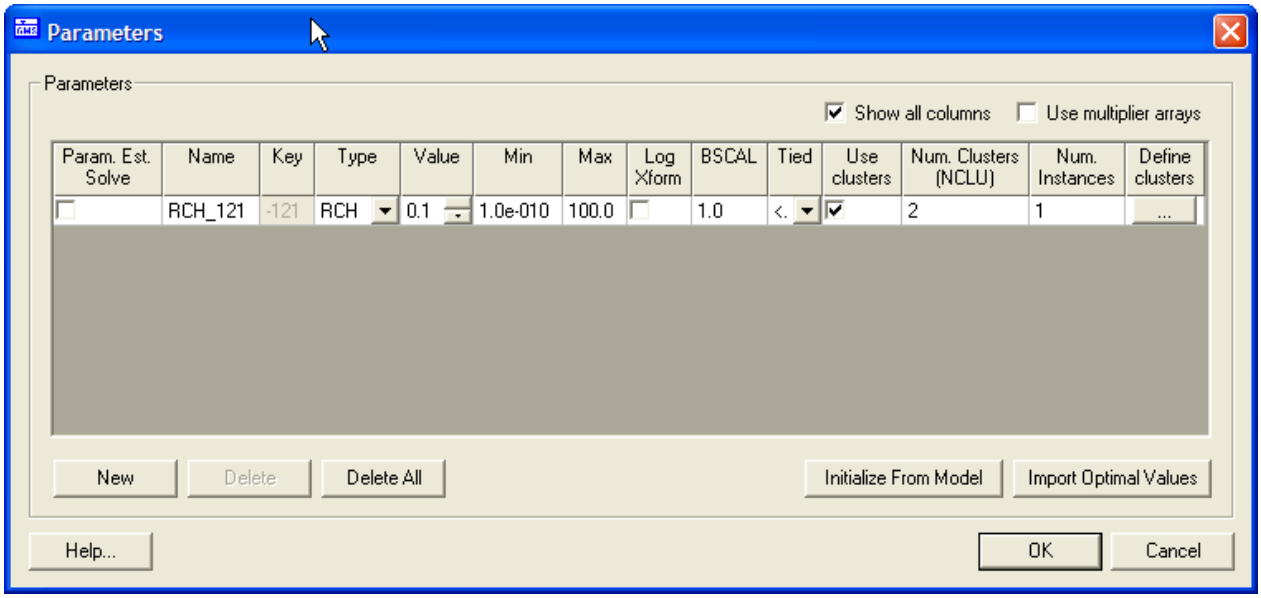
MODFLOW Parameter Clusters Dialog

The *MODFLOW Parameter Clusters* Dialog allows users to define instances for RCH, EVT, LPF, and HUF parameters that use clusters. The inputs in this dialog basically follow the necessary inputs for defining a parameter instance in a package file. You may wish to review the MODFLOW Parameter Clusters and Instances section below.

Array based parameters are defined using clusters. A cluster is a multiplier array, a zone array, and specified zone values where the parameter is applied. Multiplier arrays and zone arrays have associated data sets underneath the three dimensional grid in the Project Explorer.

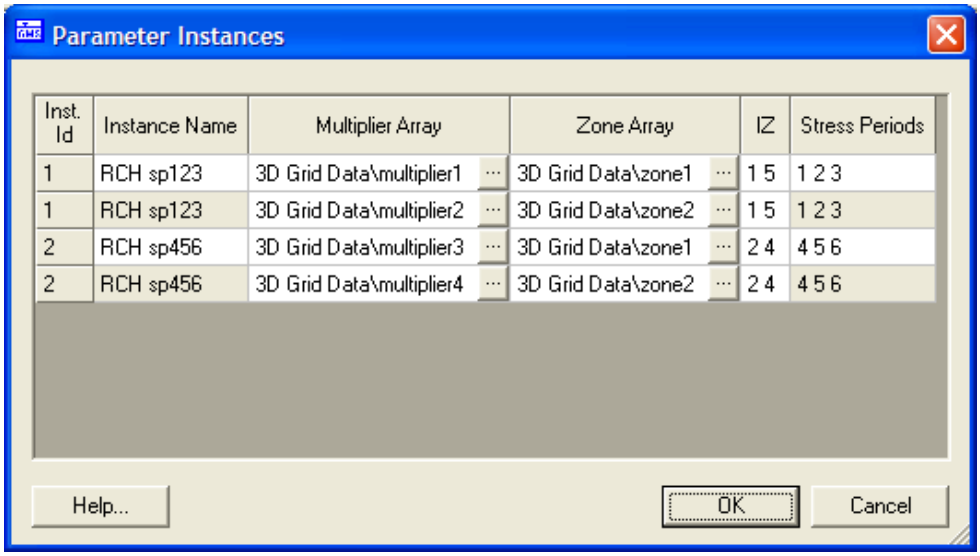
Instances are enabled for RCH and EVT parameters. Instances allow the user to define multiple sets of clusters that are all tied to one parameter. Then the user may use different instances in different stress periods in their MODFLOW model. For more information on parameter instances consult the MODFLOW documentation.

In the MODFLOW Parameters dialog the user selects the **Define clusters** option for a parameter. This will allow the user to specify the number of instances and clusters that are used to define the parameter.



In the *Parameter Instances* dialog the user defines a name for each parameter instance for RCH and EVT parameters. For LPF parameters a level is defined and for HUF parameters a hyrdogeologic unit is defined. Then for each cluster the user selects a multiplier array and a zone array. These are data sets on the 3D Grid. The user must also specify the zone values (IZ) where the parameter is to be applied in the model. If there is more than one IZ value for the cluster then the user enters the numbers with spaces between the entries. So if the parameter is to be applied where the zone array values are 1 and 3 then the user would enter "1 3". The user must also specify the stress periods where an instance is to be used. The numbers entered in the Stress Periods field should also be space delimited. This means if you want an instance to be used in stress periods 1, 2, and 5 then your entry in the Stress Periods field should be "1 2 5".

When the package is written out for MODFLOW the instances will be included in the parameter definition and used for forward runs or for parameter estimation runs.



MODFLOW Parameter Clusters and Instances

For those wanting to follow the standard MODFLOW approach to parameterization, the following explanation may be helpful. You should refer to the MODFLOW documentation for more information.

Definitions

Here is a brief review of the most important terms used when dealing with parameters.

- *Parameter* – Has a name, type, one or more clusters, and zero or more instances.
- *Parval* – The value of the parameter.
- *Cluster* – Has a multiplier array (*Mltarr*), a zone array (*Zonarr*), and zone numbers (*IZ*).
- *Mltarr* – Name of the multiplier array. Parval is multiplied by this to define the value.
- *Zonarr* – Name of the zone array used to define the cells that are associated with a parameter.
- *IZ* – Up to 10 zone numbers (separated by spaces) that define the cells that are associated with a parameter.
- *Instance* – Parameter cluster that could be used for different stress periods in a MODFLOW simulation.

Examples

Below are some examples showing different ways parameters can be used. The examples use parameters to define recharge for a forward run (not doing inverse modeling). The grid in the examples is 5 rows, 5 columns, 1 layer. The examples use the following multiplier and zone arrays.

Multiplier arrays

MULT0001

1.0	1.0	1.0	1.0	1.0
1.0	1.0	1.0	1.0	1.0
1.0	1.0	1.0	1.0	1.0
1.0	1.0	1.0	1.0	1.0
1.0	1.0	1.0	1.0	1.0

MULT0002

0.5	0.5	0.5	0.5	0.5
0.5	0.5	0.5	0.5	0.5
0.5	0.5	0.5	0.5	0.5
0.5	0.5	0.5	0.5	0.5
0.5	0.5	0.5	0.5	0.5

Zone Arrays

ZONE0001

2	2	2	0	0
2	2	2	0	0
2	2	2	0	0
0	0	0	0	0
0	0	0	0	0

ZONE0002

3	3	3	3	3
3	3	3	3	3
4	4	4	4	4
4	4	4	4	4
5	5	5	5	5

ZONE0003

0	0	0	0	0
0	0	6	7	0
0	0	6	7	0
0	0	0	0	0
0	0	0	0	0

Example 1

Steady state. 1 recharge parameter with 1 cluster, 1 instance.

Recharge file	Variables	Explanations
PARAMETER 1	PARAMETER NPRCH	One recharge parameter will be used
3 40	NRCHOP IRCHCB	Apply to highest active cell, save CCF to unit 40
Param1 RCH 0.00005 1	PARNAM PARTYP Parval NCLU	Parameter name, type, value and number of clusters
MULT0001 ZONE0001 2	Mltarr Zonarr IZ	Multiplier array, zone array, and zone number
1 1	INRECH INIRCH	One parameter used in current stress period, INIRCH (ignored)
Param1	Pname	Name of parameter used to define RECH in this stress period

Results

The resulting recharge applied to the top-most active layer would be:

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0	0	0	0	0
0	0	0	0	0

Example 2

Steady state. 2 recharge parameters:

- the first with 1 cluster, 1 instance
- the second with 2 clusters, 1 instance.

Recharge file	Variables	Explanations
PARAMETER 2	PARAMETER NPRCH	Two recharge parameters will be used
3 40	NRCHOP IRCHCB	Apply to highest active cell, save CCF to unit 40
Param1 RCH 0.00005 1	PARNAM PARTYP Parval NCLU	Parameter name, type, value and number of clusters
MULT0001 ZONE0001 2	Mltarr Zonarr IZ	Multiplier array, zone array, and zone number
Param2 RCH 0.00004 2	PARNAM PARTYP Parval NCLU	Parameter name, type, value and number of clusters
MULT0002 ZONE0002 4 5	Mltarr Zonarr IZ	Multiplier array, zone array, and zone numbers
MULT0002 ZONE0003 6 7	Mltarr Zonarr IZ	Multiplier array, zone array, and zone numbers
2 1	INRECH INIRCH	Two parameters used in current stress period, INIRCH (ignored)
Param1	Pname	Name of parameter used to define RECH in this stress period
Param2	Pname	Name of parameter used to define RECH in this stress period

Results

The intermediate results for each parameter cluster are listed below:

Param1

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0	0	0	0	0
0	0	0	0	0

Param2, cluster 1

0	0	0	0	0
0	0	0	0	0
0.00002	0.00002	0.00002	0.00002	0.00002
0.00002	0.00002	0.00002	0.00002	0.00002
0.00002	0.00002	0.00002	0.00002	0.00002

Param2, cluster 2

0	0	0	0	0
0	0	0.00002	0.00002	0
0	0	0.00002	0.00002	0
0	0	0	0	0
0	0	0	0	0

The final result after accumulating each cluster is:

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00007	0.00002	0
0.00007	0.00007	0.00009	0.00004	0.00002
0.00002	0.00002	0.00002	0.00002	0.00002
0.00002	0.00002	0.00002	0.00002	0.00002

Example 3

Transient. 1 recharge parameter with 1 cluster, 2 instances. One instance is used in the first and third stress periods, and the other is used in the second stress period.

Recharge file	Variables	Explanations
`PARAMETER 1`	`PARAMETER NPRCH`	`One recharge parameters will be used`
`3 40`	`NRCHOP IRCHCB`	`Apply to highest active cell, save CCF to unit 40`
`Param1 RCH 0.00005 1 INSTANCES 2`	`PARNAM PARTYP Parval NCLU INSTANCES NUMINST`	`Parameter name, type, value number of clusters, number of instances`
`Instance1`	`INSTNAM`	`Instance name`
`MULT0001 ZONE0001 2`	`Mltarr Zonarr IZ`	`Multiplier array, zone array, and zone number`
`Instance2`	`INSTNAM`	`Instance name`
`MULT0002 ZONE0001 2`	`Mltarr Zonarr IZ`	`Multiplier array, zone array, and zone number`
`1 1`	`INRECH INIRCH`	`One parameters used in current stress period, INIRCH (ignored)`
`Param1 Instance1`	`Pname Iname`	`Name of parameter and name of instance used in this stress period`
`1 1`	`INRECH INIRCH`	`One parameters used in current stress period, INIRCH (ignored)`
`Param1 Instance2`	`Pname Iname`	`Name of parameter and name of instance used in this stress period`
`1 1`	`INRECH INIRCH`	`One parameters used in current stress period, INIRCH (ignored)`
`Param1 Instance1`	`Pname Iname`	`Name of parameter and name of instance used in this stress period`

Results

The intermediate results for each parameter cluster are listed below:

Stress Period 1, Param1, Instance RCH_1_1

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0	0	0	0	0
0	0	0	0	0

Stress Period 2, Param1, Instance RCH_1_2

0.000025	0.000025	0.000025	0	0
0.000025	0.000025	0.000025	0	0
0.000025	0.000025	0.000025	0	0
0	0	0	0	0
0	0	0	0	0

Stress Period 3, Param1, Instance RCH_1_1

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0	0	0	0	0
0	0	0	0	0

MODFLOW Parameters - Two Methods

Caution

There are two ways to do MODFLOW parameterization in GMS:

1. The key value method **(recommended)**
2. The standard MODFLOW method

The "key value" approach is the preferred approach to defining parameters in GMS. Using key values is the only option for WEL, RIV, DRT, DRN, GHB, CHD, STR, and HFB parameters. In GMS, using the key value approach with these parameters provides the same functionality of parameter instances available in MODFLOW.

Key values can also be used with array based parameters. However, GMS does support defining array based parameters with clusters and instances (for ETS, EVT, RCH parameters).

When reading in a MODFLOW simulation that was created outside of GMS all parameters will be converted to key values so long as the parameter can be represented by key values.

Key values can not be used to represent an array based parameter if the parameter is defined using more than one cluster or more than one instance. Also, if more than one parameter uses the same zone array with a matching IZ value then the parameter can not be represented using key values. Further, if the multiplier array associated with the parameter has values other than 1.0 in the zones where the parameter is used then the parameter can not be represented with key values.

The standard MODFLOW method is considered an "advanced" feature in GMS. **Only users that understand how MODFLOW uses parameters should attempt to use this feature.**

Conductance

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Many MODFLOW boundary condition types (i.e., general head, rivers, streams, drains) include a conductance parameter. MODFLOW uses the conductance to determine the amount of water that flows in or out of the model due to the boundary condition stresses.

When using a conceptual model, the manner in which the conductance term should be computed and entered depends on whether the feature object is a polygon, arc or point. Before explaining this fully, a short review of the definition of conductance is appropriate.

Conductance formulation

Darcy's law states:

$$Q = kiA$$

where:

Q = flow rate [L^3/T]

k = hydraulic conductivity [L/T]

i = hydraulic gradient (unitless)

A = gross cross-sectional area of flow [L^2].

Darcy's law can also be expressed as:

$$Q = k\frac{\Delta H}{L}A$$

where:

ΔH = the head loss [L]

L = the length of flow [L]

Since the unknown on the right side is the head, it is convenient to group all of the other terms together and call them conductance:

$$Q = C\Delta H$$

where:

C = conductance [L^2/T]

This results in the following general definition for conductance:

$$C = \frac{k}{L}A$$

This may be represented more specifically in the following form.

$$C = \frac{k}{t}lw$$

where:

t = the thickness of the material in the direction of flow [L]

lw = the cross-sectional area perpendicular to the flow direction [L^2].

Conductance in conceptual models

In the case of a river boundary condition, the conductance is defined in MODFLOW as the hydraulic conductivity of the river bed materials divided by the vertical thickness (length of travel based on vertical flow) of the river bed materials, multiplied by the area (width times the length) of the river in the cell. The last term, area, is the hardest parameter to determine by hand since it varies from cell to cell.

Arcs

Fortunately, GMS can automatically calculate the lengths of arcs and areas of polygons. Therefore, when a conductance is entered for an arc, it should be entered in terms of **conductance per unit length**. For example, in the case of rivers, conductance should be entered as:

$$C_{arc} = \frac{\frac{k}{t}lw}{L} = \frac{k}{t}w$$

where:

Carc = conductance per unit length [(L^2/T)/L] or [L/T]

t = the thickness of the material [L]

w = the width of the material along the length of the arc [L]

When GMS applies the boundary condition from the arc to the grid cell, it automatically multiplies the entered value of conductance by the length of the arc that intersects the cell to create an accurate conductance value for the cell.

Polygons

For polygons, conductance should be entered in a **conductance per unit area** form:

$$C_{poly} = \frac{\frac{k}{t}lw}{A} = \frac{k}{t}$$

where:

Cpoly = conductance per unit area [(L^2/T)/L^2] or [1/T]

t = the thickness of the material [L]

When GMS converts the stress from a polygon to a grid cell, it automatically multiplies the entered value of conductance by the area of the cell that is covered by the polygon to create an appropriate conductance value for the cell. This restores the dimensional accuracy to the expression for conductance.

Points

When a general head, river, stream or drain attribute is assigned to an individual point, the conductance should be entered as a normal conductance value. This conductance is then directly assigned to the cell containing the point.

Parameter Factors

Beginning with GMS version 7.0, the Well, Drain, River, General head, and Stream boundary conditions have a parameter factor field that is associated with each boundary condition. The parameter factor is used with the parameter value to compute the final value of conductance for a given boundary condition (or in the case of the well package the final Q value). The parameter factor is automatically set by GMS when doing the ***Map->MODFLOW*** command. This value will be set to the length of the arc in the cell or the area of the polygon in the cell associated with the boundary condition; for a point feature the factor is set to 1.0.

MODFLOW Parameters Disclaimer

The "key value" approach is the preferred approach to defining parameters in GMS. Using key values is the only option for WEL, RIV, DRT, DRN, GHB, CHD, STR, and HFB parameters. In GMS, using the key value approach with these parameters provides the same functionality of parameter instances available in MODFLOW.

Key values can also be used with array based parameters. However, GMS does support defining array based parameters with clusters and instances (for ETS, EVT, RCH parameters).

When reading in a MODFLOW simulation that was created outside of GMS all parameters will be converted to key values so long as the parameter can be represented by key values.

Key values can not be used to represent an array based parameter if the parameter is defined using more than one cluster or more than one instance. Also, if more than one parameter uses the same zone array with a matching IZ value then the parameter can not be represented using key values. Further, if the multiplier array associated with the parameter has values other than 1.0 in the zones where the parameter is used then the parameter can not be represented with key values.

The standard MODFLOW method is considered an "advanced" feature in GMS. **Only users that understand how MODFLOW uses parameters should attempt to use this feature.**

Saving and Reading a MODFLOW Simulation

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Saving a MODFLOW Simulation

Once a MODFLOW simulation has been created and checked for potential problems with the Model Checker, the next step is to save the simulation to disk and run MODFLOW. MODFLOW simulations are saved using the ***Save*** and ***Save As*** commands in the *File* menu.

Selecting the ***Save*** command saves the GMS project including the model simulation. By default the model simulation will be saved to the same location as the GMS project. However, in the Save dialog the path for the model simulation can be specified.

A MODFLOW simulation is actually saved to a set of input files. The MODFLOW super file is used to store extra data used by GMS but not used by MODFLOW. The super file also contains the path to the name file. The names of all of the input and output files associated with a simulation are saved in the name file. When MODFLOW is launched, the name of the super file is automatically passed to the MODFLOW executable.

When a MODFLOW simulation is saved, the names of the other MODFLOW input files are automatically patterned after the name of the super file. For example, if the super file is named sampmod.mfs, the other files are named sampmod.bas, sampmod.bcf, etc.

The user can adjust the output options in the *Output Control* dialog

The version of MODFLOW 2000 that ships with the GMS software has been modified to be able to read and write input data stored in HDF5 files. For more information see MODFLOW with HDF5.

Reading a MODFLOW Simulation

Once a MODFLOW simulation has been saved by GMS using the ***Save*** command in *File* menu, the entire simulation can be read back into GMS using the ***Open*** command in the *File* menu.

MODFLOW simulations can be read in individually (*.mfs) or as part of a project (*.gpr). They can also be imported by using the MODFLOW name file. The extension for the name file must be changed to *.mfn.

Reading a Completed MODFLOW Solution

When you are using the Model Wrapper to control your MODFLOW run, you can use the **Read MODFLOW solution** toggle at the bottom of the dialog to automatically read in the corresponding solution.

You can also read in solution files generated by MODFLOW using the ***Read Solution*** command in the *MODFLOW* menu. This command brings up the a file browser and prompts you to choose a MODFLOW super file (pre GMS 7.0) or a MODFLOW name file (GMS 7.0+).

Output Control

The Output Control dialog is accessed from the *MODFLOW* menu. The Output Control dialog contains the following options:

Output Interval Options

The Output Interval section lets you choose which files you want to output and when you want to output the results. By default, GMS instructs MODFLOW to output heads and volume data to both data files and text output files. Drawdown is not output by default.

Global Output File

Choose this option to use the global output file (*.glo). This file contains information about the input files and, during an inverse run using the PES process, will contain information about each iteration. Either the global file, list file, or both must be used with MODFLOW.

List Output File

Choose this option to use the list output file (*.out). This file contains information about the current MODFLOW run. This file will be overwritten during an inverse run using the PES process for each inverse iteration.

*.hff File For Transport

Select this option to output the head and flow file (*.hff) used by MT3DMS to generate the flow fields for transport modeling.

Save Arrays Using Binary Files

Choose this option to save all 2D arrays used as MODFLOW input as binary arrays in external files. If this option is not selected, the 2D arrays are written within the respective files in ASCII format. The binary option results in more files, but uses less disk space. There is also a time savings during running for MODFLOW models with large grids.

Enable saving of computed flows for all source/sink objects

This option allows GMS to create a flow observation for all sources/sinks that were created using the ***Map->MODFLOW*** command so that the user can see the computed flux through the group of boundary conditions. If the user is doing a transient simulation then having this option may cause MODFLOW to crash because MODFLOW attempts to allocate more memory than any of us have available. Turning off this option in this situation is the best course of action.

Importing MODFLOW Files

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

GMS imports standard MODFLOW 88, 96, 2000, 2005[1], and MODFLOW-NWT[2] files. GMS supports saving of MODFLOW 2000, 2005, and MODFLOW-NWT so other versions will be converted to MODFLOW 2000, 2005, or NWT.

If GMS created the MODFLOW simulation then you should open the corresponding GMS project file (*.gpr). If there is no GMS project file, you can import the MODFLOW simulation into GMS as described below. GMS will create a new copy of the imported MODFLOW simulation in its own modified MODFLOW file format. Click here for more information on the modified MODFLOW file format.

Native MODFLOW files (ascii/text) can be exported from GMS. Click here for more information on exporting MODFLOW files.

How to import a MODFLOW model into GMS

You can follow these steps when importing a MODFLOW simulation into GMS.

1. Were the MODFLOW files created by GMS?
   - Yes

     If the files were created by GMS, and you have the GMS project file (.gpr), you should just read that into GMS using the standard ***File|Open*** command. If you don't have the .gpr file, you should import the MODFLOW model into GMS by opening the "super file" (*.mfs) using the standard ***File|Open*** command. The super file is a non-standard file that GMS creates along with the standard MODFLOW files. If you don't have a super file, proceed to step 2 below.
   - No

     Proceed to step 2 below.
   - Don't know

     If you have a *.gpr file or a *.mfs file, then it's almost certain that the files were created by GMS. If you don't have these files, proceed to step 2 below.
2. Determine whether your model is MODFLOW 88, 96 or 2000. If you are unsure, refer to the section below entitled Determining the MODFLOW version which describes each one.
   - MODFLOW 88

     Read the basic package file into GMS (using the standard *File|Open* dialog). GMS will attempt to import all the other files.
   - MODFLOW 96

     Read the name file.
   - MODFLOW 2000

     Read the name file.
   - MODFLOW 2005

     Read the name file.
   - MODFLOW-NWT

     Read the name file.

Troubleshooting

If you are having trouble reading the files into GMS, first verify that MODFLOW can read the files by launching MODFLOW at a command prompt and giving it the name of the file to read. If MODFLOW can read the files but GMS cannot, it may be because you are attempting to import packages which GMS does not support; see MODFLOW Packages Supported in GMS for more information.

Contact tech support [1] for additional help.

Determining the MODFLOW version

MODFLOW 88

To tell if a file is MODFLOW 88 open up the basic package file as a text file. GMS uses *.bas as the extension for this file. The file should appear something like the figure below.

```
Heading 1
Heading 2
          3         23         17          1          4
 11 13 14 15  0  0 16  0  0  0  0 10 12  0  0  0  0  0  0  0  0 29  0  0
          0          0
          1          1                (17I3)          0
  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1
  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1
  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1
  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1  1
```

For a MODFLOW 88 file the third line of the basic package file will include an IUNIT array with 24 slots. There may also be an IUNIT array with only 12 slots, if so just ok the warning GMS gives. These slots include the unit numbers for packages included in the file. Every number must be unique to its package. If the number does not correspond to a file then the file is an external array which must be matched by the user. If the basic package file includes an IUNIT array as described it is a MODFLOW 88 file.

To import the MODFLOW 88 file locate the *.mfs file. If there is no *.mfs file then select the *.bas file. Then simply use the ***File\Open*** command to read the file into GMS.

More about MODFLOW 88 files

MODFLOW 88 files have no name (*.nam or *.mfn) file.

GMS makes a few assumptions when reading in MODFLOW 88 files:

1. All files must have the same prefix.
    - Example: If the files are named bas.dat, drain.dat, and river.dat they all must be converted to run1.bas, run1.drn, and run1.riv
2. All files use the standard GMS suffixes as shown in the table below.
    - If these are not the suffixes in use, you will need to rename the files.
3. IUNIT slots must be standard with standard ID #'s according to MODFLOW documentation.
4. It is up to the user to know what external arrays go to which MODFLOW files. Otherwise the files will not be properly read in.
    - GMS can import external arrays but is unable to import external binary arrays
5. MODFLOW 88 files use a fixed format.

MODFLOW 88 IUNIT Array Positions and Packages

1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20
BCF	WEL	DRN	RIV	EVT		GHB	RCH	SIP	DE4	SOR	OC	PCG	GFD		HFB	RES	STR	IBS	CHD

Standard GMS Extensions for MODFLOW 88 files

Package	Extension
BASIC	*.bas
OUTPUT CONTROL	*.oc
BCF	*.bcf
RIVER	*.riv
DRAIN	*.drn
WELL	*.wel
GENERAL HEAD BOUNDARY	*.ghb
STREAM	*.str
RECHARGE	*.rch
EVAPOTRANSPIRATION	*.evt
STRONGLY IMPLICIT PROCEDURE	*.sip
SLICE-SUCCESSIVE OVERRELAXATION	*.sor

MODFLOW 96

To tell if model files are in MODFLOW 96 format, open up the basic package file as a text file. For a MODFLOW 96 file the third line of the *.bas file will NOT include an IUNIT array. Instead the third line will say FREE, which means that the data in the file is in free format and each data entry is separated with a space. A 96 file may not contain the FREE line. Instead the line may be blank and the file may be fixed format. This file can be read in just the same using the name file.

```
Heading 1
Heading 2
         3         23         17          1          4
FREE
          0          0
INTERNAL 1 (free) 0 "Ibound Layer 1"
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
```

If the basic package file says FREE as described it is a MODFLOW 96 file. A name file (typically with a *.mfn or *.nam extension) should also exist. If one does not it must be created as described below.

To open a MODFLOW 96 file locate the name file and use the ***File|Open*** command to read the file into GMS.

More about MODFLOW 96 files

MODFLOW 96 files use a name file to identify the packages files. This replaces the IUNIT array used in MODFLOW 88. The advantage of it is that the files do not need to all have the same prefix in their name. The name file is laid out as shown below. In the first column the card name is given (see MODFLOW documentation for more information). In the second colum the IUNIT number is given. This number must be unique from all other numbers. The third column gives the file name and as stated does not need to have the same prefix for every package.

Sample *.mfn file:

```
LIST   26 "run1.out"
BAS    1 "run1.bas"
BCF   11 "run1.bcf"
OC   10 "run1.oc"
DATA(BINARY)  30 "run1.hed"
DATA(BINARY)   40 "run140.ccf"
PCG   12 "run1.pcg"
RIV   15 "run1.riv"
WEL   13 "run1.wel"
RCH   20 "run1.rch"
MT3D   29 "run1.hff"
```

MODFLOW 2000, 2005[1], & MODFLOW-NWT[2]

Like the MODFLOW 96 files, MODFLOW 2000, 2005 and MODFLOW-NWT use name files rather than an IUNIT array. To know if a file is a MODFLOW 2000, 2005, or MODFLOW-NWT file look for a discretization file (*.dis). If a *.dis file exists then use the ***File\Open*** command and select the name file (*.nam or *.mfn) to read the simulation into GMS.

Sample *.mfn file:

```
# MF2K NAME file
#
# Output Files
GLOBAL 1 "easttex.glo"
LIST 2 "easttex.out"
DATA(BINARY) 30 "easttex.hed"
DATA(BINARY) 40 "easttex.ccf"
LMT6 18 "easttex.lmt"
#
# Obs-Sen-Pes Process Input Files
OBS 50 "easttex.obs"
DROB 54 "easttex.drob"
CHOB 55 "easttex.chob"
ASP 71 "easttex.asp"
#
# Global Input Files
DIS 19 "easttex.dis"
#
# Flow Process Input Files
BAS6 3 "easttex.ba6"
LPF 4 "easttex.lpf"
```

```
OC 15 "easttex.oc"
RCH 16 "easttex.rch"
WEL 9 "easttex.wel"
DRN 10 "easttex.drn"
CHD 13 "easttex.chd"
PCG 14 "easttex.pcg"
```

Changes in GMS 7.0

When GMS 7.0 or later reads a MODFLOW simulation it checks if the simulation files are in the modified GMS format. MODFLOW files created by GMS 6.5 or a later use this format. If the simulation is not in the GMS format then GMS converts it into this format.

In versions of GMS prior to 7.0, GMS would read the MODFLOW files and do the conversion. Starting with version 7.0, GMS uses a modified version of MODFLOW to read the input files and do the conversion. GMS then reads the converted files. By using MODFLOW to read MODFLOW files and do the conversion, GMS can read whatever MODFLOW can read. Conversely, if GMS cannot read it, MODFLOW probably can't either.

Conversion Steps

Depending on the version of your model, GMS will perform all or some of the following steps to convert the model into the standard GMS format.

1. **Convert MODFLOW 88 to MODFLOW 96**. If importing a MODFLOW 88 model, GMS creates a MODFLOW 96 Name file by examining the Basic package file or the GMS super file. The name file is put in the same directory where the Basic package file or super file is found. In creating the MODFLOW 96 name file, GMS assumes the unit numbers in the IUNIT array in the Basic package file are ordered according to the standard MODFLOW 88 order (as found in the MODFLOW 88 source code). GMS uses the lowest unit number not used in the IUNIT array for the List file added to the name file. If external arrays are being used, the unit number GMS uses for the List file may be the same as one used for an external array.
2. **Convert MODFLOW 96 to MODFLOW 2000**. After creating the MODFLOW 96 Name file, GMS calls mf96to2k.exe, a USGS program distributed with MODFLOW 2000 which converts MODFLOW 96 models to MODFLOW 2000. This creates some new files in the same directory as the basic package file: name_MF2k.bas, name_MF2K.bcf, and name_MF2K.nam (where "name" is the name of the MODFLOW 96 name file).
3. **Translate MODFLOW 2000 to GMS H5 Format**. GMS then calls the MODFLOW Translator to translate the simulation into GMS H5 format. This creates an H5 file in the temp directory. The user must save the GMS project in order to have a GMS H5 formatted MODFLOW simulation on disk, otherwise the temp file will be discarded.

mf96to2k

Some modifications were made to mf96to2k.exe with regards to layer elevations and confining beds so that the program could be run in a non-interactive batch mode.

Grid Elevations

When mf96to2k.exe converts a MODFLOW 88/96 simulation to a MODFLOW 2000 simulation the program creates a discretization file (DIS). This file defines the layer elevations for the model grid as well as the stress periods. In MODFLOW 88/96 a model could be defined without the need for explicitly defining the top and bottom elevations of the model grid. If the old model has grid elevations defined in the BCF file then those elevations are preserved in the new MODFLOW 2000 DIS file. Where elevations are not defined in the old model then mf96to2k.exe would prompt the user to enter a constant elevation for the particular layer. When mf96to2k.exe is run by GMS these layer

values are automatically set. After the model is read into GMS the user may need to edit these values.

Layer Confining Beds (LAYCBD)

When mf96to2k.exe converts a MODFLOW 88/96 model it also asks the user if a layer confining bed exists beneath each of the grid layers. When mf96to2k.exe is run by GMS no confining beds will be defined beneath the model layer. If the user wishes to include confining beds these can be edited in the *Global/Basic* dialog under the *MODFLOW* menu.

Files generated by other software

Other software such as Groundwater Vistas, Visual MODFLOW, and PM Win use their own file formats. However they do write out native MODFLOW files which GMS can read in. To use them make sure that the other software is saving out the files properly and then follow the steps above for the proper version of MODFLOW files.

Groundwater Vistas

To export MODFLOW files from Groundwater Vistas:

1. Open the simulation in Groundwater Vistas
    1. Select “File | Open”
    2. Navigate to and select the “*.gwv” file, then select “Open”
2. Export the native MODFLOW files from Groundwater Vistas
    1. Select “Model | MODFLOW (or MODFLOW 2000) | Create Datasets”
3. Import the name file into GMS
    1. Select “File | Open”, the “Open” macro, or just drag and drop the *.nam file in the GMS main screen.

PM Win

1. Select "Models | MODFLOW | Run"
2. Prior to MODFLOW running the model inputs will be generated.

Visual MODFLOW

Visual MODFLOW files can not be read into GMS. To use Visual MODFLOW files, you'll need to run MODFLOW from within Visual MODFLOW, and then modify the *.mfi file. See below for details.

1. Run MODFLOW from Visual MODFLOW
    1. Open the Visual MODFLOW project (vmf file if using the latest version of Visual MODFLOW) in Visual MODFLOW
    2. Select “Run” in the top menu
    3. Select “Run” again from the top menu
    4. Select “MODFLOW 2000” in the “Engines to Run” dialog, then select “Translate & Run”
    5. Select the “Close” button in the “VMEngine” window when the model is finished running
2. Modify and rename the “*.mfi” file
    1. Go to the directory where the simulation is saved on your computer
    2. Create a copy of the “*.mfi” file. Change the extension of the copy to “*.mfn”
    3. Open the *.mfn file in a text editor
    4. Comment out packages that are not supported by the USGS release of MODFLOW (for example, the “WHS” and “NDC” lines)^
    5. If necessary, modify the directory that each MODFLOW file is referencing. This is necessary if the Visual MODFLOW files were not created on the computer you are on

6. Save the *.mfn file

3. Open the *.mfn file in GMS
   1. Select “File | Open”, the “Open” macro, or just drag and drop the *.mfn file in the GMS main screen
   2. The “MODFLOW Translator” will most likely appear. Select “OK” after selecting the appropriate version of MODFLOW
   3. Select “Done” when the MODFLOW Translator is finished
   4. A dialog will appear saying that a supported solver was not found in the name file and that the PCG solver has been added. This is because the solver that Visual MODFLOW uses is not compatible with the USGS version of MODFLOW, and so GMS does not use it either. Select “OK”

^Note: to comment out a package, place a pound/number sign in front of a line. See the Online Guide to MODFLOW [3] for packages that the USGS version of MODFLOW supports.

USGS MODFLOW Documentation

Here are some links to MODFLOW documentation that may be helpful.

MODFLOW 88	MODFLOW 88 (I) [5], MODFLOW 88 (II) [6]
MODFLOW 96	MODFLOW 96 (I) [7], MODFLOW 96 (II) [8]
MODFLOW 2000	MF2K Flow [2], MF2K Obs-Sen-Pes [13], HUF [10], LMG [11], LMT [12], MF2K Calibration [9]
MODFLOW 2005	MF2005 Flow Process [3]
MODFLOW-NWT	MODFLOW-NWT, A Newton Formulation for MODFLOW-2005 [4]

Notes

[1] Supported starting at GMS version 8.0

[2] Supported starting at GMS version 8.2

[3] http://water.usgs.gov/nrp/gwsoftware/modflow2000/MFDOC/index.html?introduction.htm

MODFLOW with HDF5

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The versions of MODFLOW 2000, MODFLOW 2005[1], and SEAWAT[1] that ship with the GMS software have been modified to be able to read input data stored in HDF5 files. GMS saves some MODFLOW data to an HDF5 file that is saved with the other MODFLOW files. Some MODFLOW files will have references to the HDF5 file. For more information about HDF5, click here[2].

Advantages to HDF5

The HDF5 file format is a cross platform binary format for storing scientific data. One advantage of using HDF5 is that the data is stored in binary format so the disk reads and writes are much faster than with ASCII files (although not nearly as elegant to read). You can view the data stored in an HDF5 file by using HDFView [3] (or some other available viewer).

Another advantage of using the HDF5 libary with MODFLOW package data and array data is that HDF5 will compress the data. For example, if you have a transient simulation using the river package the typical line in a river package file would look like this:

[K] [I] [J] [STAGE] [CONDUCTANCE] [ELEVATION]

1 3 4 10.2 5.8 9.5

Then during the next stress period typically the stage would change. The line in the file would look like this:

1 3 4 10.8 5.8 9.5

So we repeated all of the data except that 10.2 changed to 10.8. With compression those repeated values do not take up much space on disk so that the resulting file is smaller.

In one example a river file that was 437 MB was saved to HDF5 format with a compression level of 1; the resulting file was 14 MB. In another example an HDF5 file that contained drain and general head data was 8 MB while the drain file was 253 MB and the general head 141 MB. Not only do the files compress extremely well but the read/write times for the HDF5 file are much faster than the ASCII files.

Changes to MODFLOW 2000 & 2005[1] code

The following subroutines in ult6.f were modified: U2DREL, U2DINT, and ULSTRD. The stream package reading routines were also modified in the str6.f file.

ult.f modifications

U2DREL and U2DINT were modified so that if the key word HDF5 was encountered when attempting to read an array then an external procedure is called to read in the data. We have added support for the following:

Explanation of variables

HDF5	a card to indicate that this is read from an HDF5 file
CNSTNT	the multiplier for the array
IPRN	print flag for the output control for MODFLOW
"FNAME"	the HDF5 file where the array is stored
"pathInFile"	path to the data set in the HDF5 file
nDim	number of dimensions that the data set has (this is 1, 2, 3)
start1	the index (NOTE: these are 0 based **not** 1) for the starting point to read the data set in the first dimension
nToRead1	the number of values to read in the first dimension
start2	the index for the starting point to read the data set in the second dimension
nToRead2	the number of values to read in the second dimension
start3	the index for the starting point to read the data set in the third dimension
nToRead3	the number of values to read in the third dimension

example: "HDF5 1.0 0 "input.h5" "Recharge/07. Property" 3 0 1 0 132 0 1"

This type of formatting will work for any HDF5 data set that has 3 dimensions or less and the data set can be a 4 byte float or an 8 byte double. The other available format that can be used with the HDF5 key word for the array reading utilities is the following:

HDF5 CONSTANT CNSTNT

example: HDF5 CONSTANT 3.0

This type of formating will assign a constant value to the array. This type of formatting may seem unnecessary. However, the constant identifier can be useful when using parameters to define arrays. Parameters will be explained in another section.

ULSTRD was also modified to read data from an HDF file. This format is specific to the types of HDF5 files that GMS creates.

GMS_HDF5_01 "FNAME" "pathInFile" SP

Explanation of variables

GMS_HDF5_01	a card to indicate that this is read from an HDF5 file
"FNAME"	the HDF5 file where the array is stored
"pathInFile"	path to the group in the HDF5 file where all of the data for the boundary condition is stored
SP	stress period number

Here is an example of a regular modflow drain file and a drain file with the HDF5 modifications.

Comparison of MODFLOW Drain files

Traditional Drain file	Drain file with HDF5
3 40 AUX IFACE AUX CONDFACT AUX CELLGRP 3 0 1 3 2 19.0 10.0 6 1.0 -1 1 4 3 19.0 10.0 4 1.0 -1 1 5 4 19.0 10.0 5 1.0 -1 3 0 1 3 2 18.9 11.0 6 1.0 -1 1 4 3 18.9 11.0 4 1.0 -1 1 5 4 18.9 11.0 5 1.0 -1 3 0 1 3 2 19.0 10.0 6 1.0 -1 1 4 3 19.0 10.0 4 1.0 -1 1 5 4 19.0 10.0 5 1.0 -1	#GMS_HDF5_01 3 40 AUX IFACE AUX CONDFACT AUX CELLGRP 3 0 0 GMS_HDF5_01 "sg_t_pest_65.h5" "Drain" 1 3 0 0 GMS_HDF5_01 "sg_t_pest_65.h5" "Drain" 2 3 0 0 GMS_HDF5_01 "sg_t_pest_65.h5" "Drain" 3

str6.f modifications

Since the STR package does not use the ULSTRD utility the str6.f file had to be modified to read the same data that ULSTRD was modified to read: **GMS_HDF5_01 "FNAME" "pathInFile" SP**.

Parameters

MODFLOW 2000 introduced the concept of parameters as native input to MODFLOW. Starting with version 6.5, GMS no longer writes out native MODFLOW parameters (except when clusters are defined) and the use of the MODFLOW PES process is not available from within GMS. Even when native MODFLOW files are exported from GMS, native MODFLOW parameters are not used (except when clusters are defined) and instead the parameter starting values are written. However, GMS still supports PEST and users may create parameters in the GMS interface as they have been able to do previously. Parameters are created in GMS by assigning a "key" value (usually a negative number) to a MODFLOW input. When MODFLOW is running and calls our external routine to read in the HDF5 a check is made in the data that is read from the HDF5 file. If one of the parameter "key" values is found in the data then the parameter value is substituted into the array or list.

By adopting this approach it was much easier to support transient parameters and very large sets of pilot points.

Also, adopting this approach allows GMS users to use parameters with the BCF package. This feature is not available with the MODFLOW PES process.

Pilot Points

In previous versions of GMS, pilot points were supported by using the multiplier arrays in MODFLOW. Beginning with version 6.5 the pilot point interpolation takes place with in our external routine that is called by MODFLOW. For example, if MODFLOW calls our routine to read an HK array and that array has parameter *key* values that are associated with a parameter that is defined using pilot points then our routine will perform the pilot point interpolation and substitute the appropriate value. This includes any log interpolation that the user has specified.

Reading MODFLOW files which include HDF5 data

When GMS writes out the MODFLOW files for the list based packages, the first item written to the file is #GMS_HDF5_01. When GMS reads these MODFLOW files with this special comment, it does not read any more of the package file and the data from the *.h5 file will be used.

When GMS saves the RCH or EVT package with parameter that are defined with cluster (and instances). Then a different special comment is written to the top of the file: #GMS_PARAM_CLUSTERS. If GMS is reading a project file that includes a MODFLOW simulation then when this special comment is encountered GMS will not read any more of the package file. However, if a user is reading a MODFLOW model (not a project file) then the rest of this package file will be read and new data sets will be created on the 3D grid that are used to define the clusters used by the parameters.

Exporting Native MODFLOW Files

To export native MODFLOW ASCII files from GMS 6.5 (or later), use the following steps:

1. Open a project in GMS (6.5 or later).
2. In the 3D Grid Module, select ***MODFLOW\Export Native MF2K text...***.
3. In the window that appears, double-click the Name file of the MODFLOW model that you want to convert to ASCII format.
4. A DOS prompt will appear. Press the Enter key when the executable is finished.
5. A folder called **Out_Mf2k** is created in the same folder as the name file you selected. **Out_Mf2k** contains all the MODFLOW input files in ASCII format.

If the DOS prompt does not appear after selecting the name file, then go to Edit|Preferences... and select Models. Verify that the **Gms2Mf2k** executable (the application that converts MODFLOW files from HDF5 to ASCII format) has the correct path next to it. If it doesn't, select ***Browse...*** and locate the executable. It will probably be located at **C:\Program Files\GMS 8.1\models\mf2k\Gms2Mf2k.exe**.

Notes

[1] Supported starting at GMS version 8.0

[2] http://hdf.ncsa.uiuc.edu/HDF5/

[3] http://hdf.ncsa.uiuc.edu/hdf-java-html/hdfview/index.html

Unsupported MODFLOW Features

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

MODFLOW Versions

The supported MODFLOW versions include:

- MODFLOW 2005 (Supported starting at GMS version 8.0. Read/Write)
- MODFLOW 2000 (Read/Write)
- MODFLOW 96 (Read and convert to MODFLOW 2000. No write)
- MODFLOW 88 (Read and convert to MODFLOW 2000. No write)

The unsupported MODFLOW versions include:

- MF2K-GWM
- MF2K-VSF
- MF2K-FMP

Processes

For GMS 6.5 and 7.0 the following processes are not supported:

- PES (This process is not needed since GMS uses PEST for parameter estimation)
- SEN (This process is not needed since GMS uses PEST for parameter estimation)

Packages

- See Packages Supported in GMS. All other packages are unsupported (but we are always working on adding support for more packages). For any package not currently supported in GMS, the file associated with that package will be ignored when the model is imported. However, GMS keeps track of unsupported packages it encounters when reading the name file and, when the project is saved, GMS lists the unsupported packages in the name file and copies the unsupported package files from the old directory to the new directory.

Options

The following MODFLOW options are not supported:

- XSECTION
  - GMS does not support models that use the XSECTION option on line 1 of the BA6 file.
- Quasi-3D confining bed (GMS 6.5.x and before)
- File formats: GMS 6.5.x (and earlier versions) can read most regularly formatted files but MODFLOW supports more formatting options than GMS does. Beginning with version 7.0, MODFLOW is used to import non-GMS MODFLOW simulations (thus, all MODFLOW formats can be imported).

Parameters

MODFLOW 2000 introduced the concept of parameters as native input to MODFLOW. Starting with version 6.5, GMS no longer writes out native MODFLOW parameters (except when clusters are defined) and the use of the MODFLOW PES process is not available from within GMS. Even when native MODFLOW files are exported from GMS, native MODFLOW parameters are not used (except when clusters are defined) and instead the parameter starting values are written. However, GMS still supports PEST and users may create parameters in the GMS interface as they have been able to do previously. Parameters are created in GMS by assigning a "key" value (usually a negative number) to a MODFLOW input. When MODFLOW is running and calls our external routine to read in the HDF5 a check is made in the data that is read from the HDF5 file. If one of the parameter "key" values is found in the data then the parameter value is substituted into the array or list.

By adopting this approach it was much easier to support transient parameters and very large sets of pilot points.

Also, adopting this approach allows GMS users to use parameters with the BCF package. This feature is not available with the MODFLOW PES process. Other unsupported parameter options:

- HFB, SFR (GMS 7.0.x and before)
- STR, CHD (GMS 6.5.x and before)
- Multiple clusters for array based parameters (GMS 6.5.x and before)
- Parameter Instances (GMS 6.5.x and before)

Interpolate to MODFLOW Layers

In this dialog the user can select various 2D scatter point data sets and associate them with MODFLOW input data. GMS will then interpolate from the 2D scatter point data to the MODFLOW input data. This is often useful to define the layer elevations of a MODFLOW model. Click here for more information on MODFLOW layer elevations.

The user associates 2D scatter point data with MODFLOW data by selecting a data set in the list on the left of the dialog and selecting an item from the MODFLOW data list on the right and then selecting the **Map** button. The association between the data sets and the MODFLOW data is shown in the spread sheet labeled *Data set => MODFLOW data*.

The *MODFLOW data* group box contains options to allow the user to pick which MODFLOW data they would like to interpolate to. *Elevations* refers to the top and bottom elevations of the 3D grid. *Heads* refers to the starting heads. *Flow package data* refers to the various input arrays for the LPF or BCF packages (HK, VK...). *Recharge* refers to the recharge rate specified in the RCH package. *CHD BCs* refers to the starting and ending heads associated with existing CHD boundary conditions in the MODFLOW model.

The **Automap** button will try to automatically map the data sets to the MODFLOW data by matching names and layer numbers. The Unmap button is used unmap the selected item(s) in the spread sheet. The **Unmap All** button is self explanatory.

The Interpolation Options button will bring up the *2D Interpolation Options* dialog.

Transient Interpolation

If a 2D scatter point data set is transient and the user is interpolating to MODFLOW data that does not vary with time then the user will select a time step from that data set to interpolate to the MODFLOW data. When the user has selected a time step the **Apply Selected Time To All Transient Data Sets** button will undim. This button allows the user to set the selected time of all of the transient data sets to be the same as the currently selected time.

If the current MODFLOW model is transient and the user is interpolating to Recharge then interpolation will occur for each stress period defined in the MODFLOW model. For a particular stress period, the closest data set time step whose time is less than the beginning of the stress period is used. Figure 1 illustrates this process.

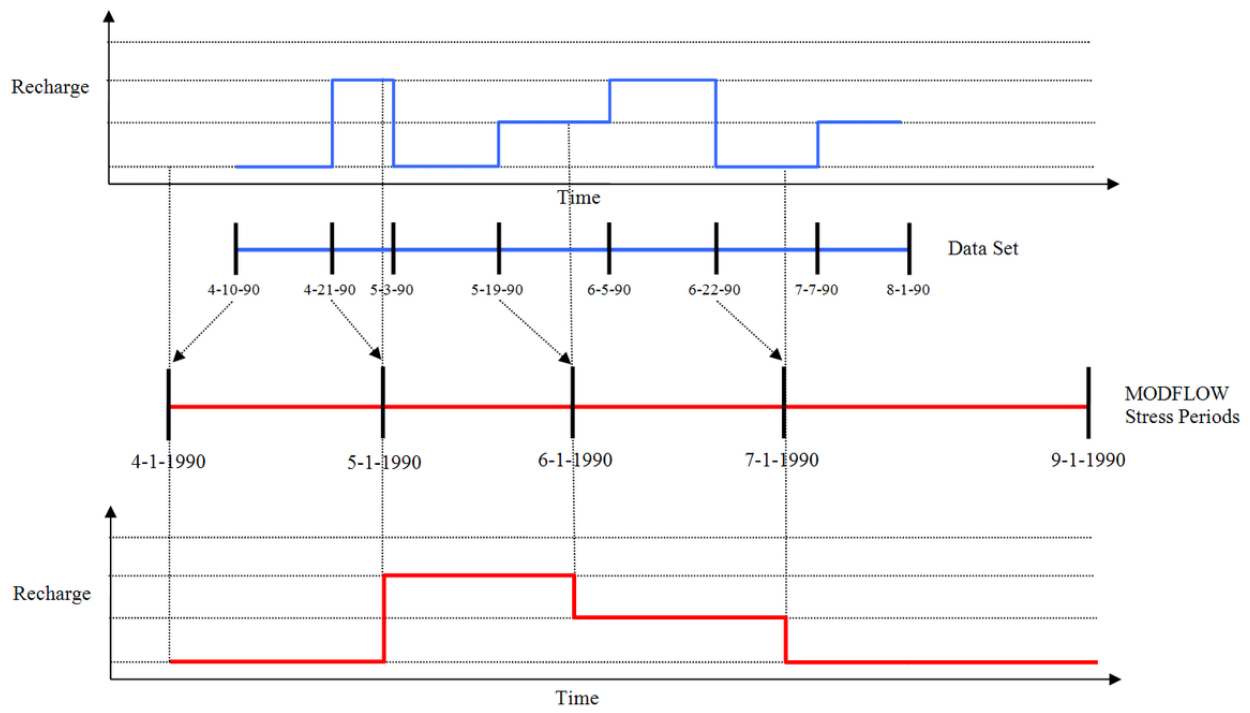


Figure 1: Transient interpolation of Recharge

If the current MODFLOW model is transient and the user is interpolating to CHD BCs then interpolation will occur at the beginning and the end of each stress period (because a start head and an end head are specified for each CHD BC). If there is no data set time step that matches the beginning or the end of the stress period then GMS will linearly interpolate between data set time steps. Figure 2 illustrates this process.

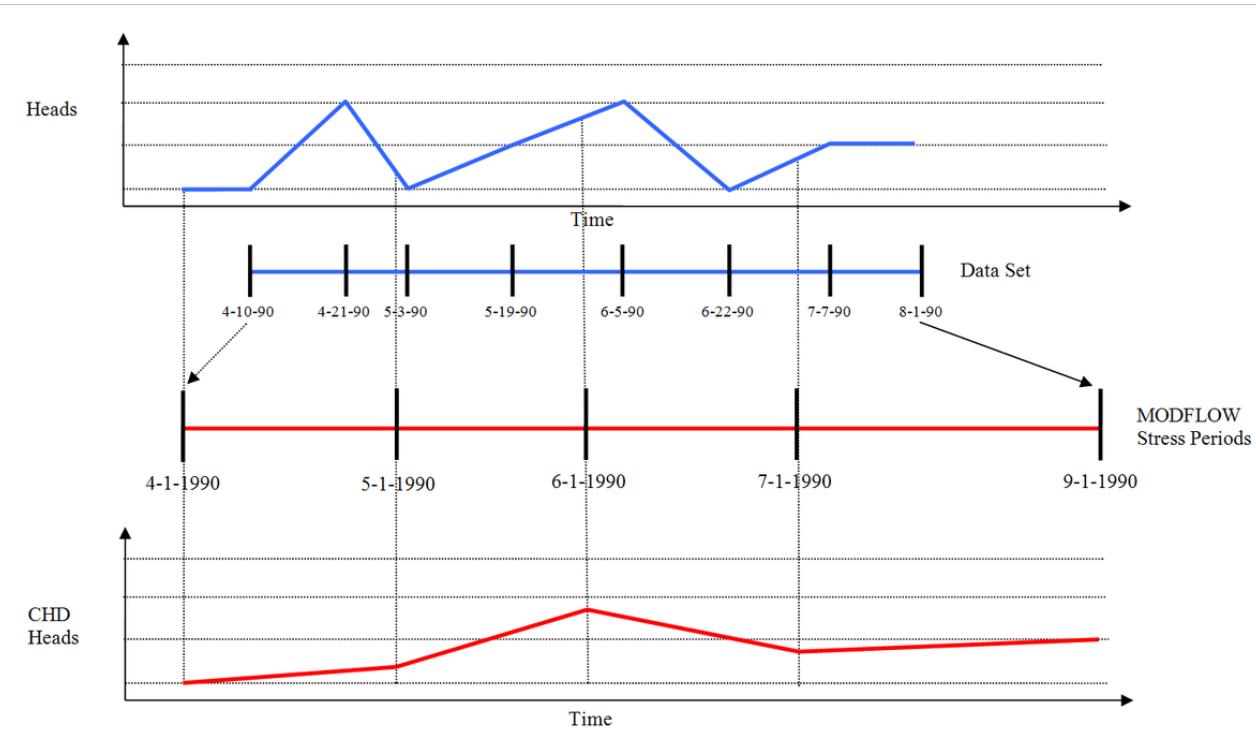


Figure 2: Transient interpolation of CHD BCs

If the current MODFLOW model is transient and the user is interpolating to GHB BCs then interpolation will occur at the beginning of each stress period. If there is no data set time step that matches the beginning or the end of the

stress period then GMS will linearly interpolate between data set time steps. Figure 3 illustrates this process.

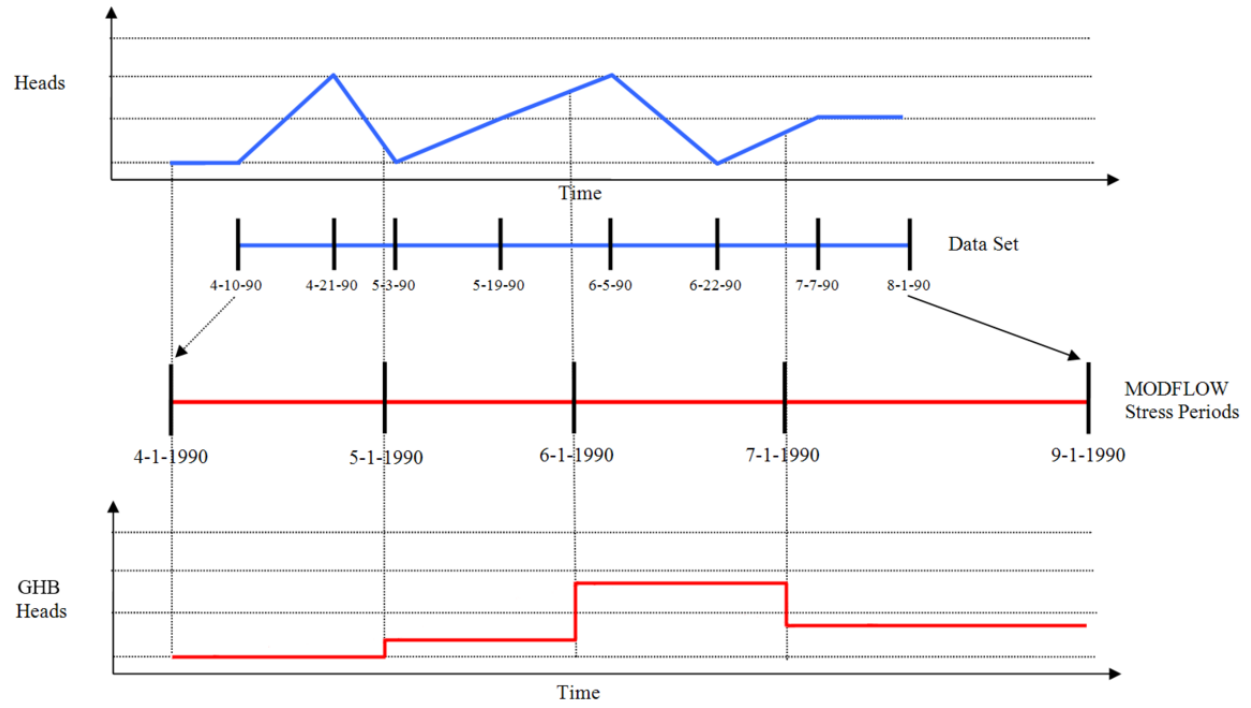


Figure 2: Transient interpolation of GHB BCs

MODFLOW Layers to 2D scatter points

This dialog allows the user to create a scatter point set and data sets of the current MODFLOW simulation.

The user can limit the number of scatter points created by turning on the **Only create scatter points within selected coverage option** and selecting the appropriate coverage. This is often used when converting between a regional model and a local scale model.

Layer Subdivision

The layers of the current MODFLOW simulation can be subdivided by specifying the *Number of local model layers* for each layer in the current 3D grid.

The user can select which MODFLOW data sets to create by turning off/on the toggles below the *Create data sets of* text. Data sets for layer elevations, flow package data (HK, HANI, VK, SY, SS...), Recharge, and Heads can be created. The user must select a 3D grid data set in order to create a data set for the Heads (most often this will be the MODFLOW solution from the regional model).

The Recharge and Head data sets can be transient if the regional model was transient. The user can also select a Start and End time to limit the number of time steps for these data sets.

Bias Layer 1

An option to bias the thickness of the new layer 1 is also available. This can be useful if equally subdividing layer 1 of the regional model would result in cells where the elevation would be above the computed heads. Thus, the user can choose to bias the thickness of the new layer 1 to ensure that the top layer in the new model will not be dry.

MODFLOW Source/Sink Dialog

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The Sources/Sinks dialog allows the user to edit both point sources/sinks and areal sources/sinks.

Point Sources/Sinks

The MODFLOW stresses can be categorized as point sources/sinks and areal sources/sinks. The point sources/sinks include rivers, wells, drains, and general head. Point sources/sinks are not assigned using arrays. Rather, they are associated with individual cells. Two methods are used to define point source/sink data. The simplest method is to define them as part of a MODFLOW conceptual model defined in the Map module. Another method is to assign them by selecting a set of cells and using the Sources/Sinks command.

The Sources/Sinks command is used to both assign and edit river, drain, general head, and well type sources/sinks. Before selecting the Cell Sources/Sinks command, a set of cells should be selected using the cell selection tools. Once the command is selected, the Point Sources/Sinks dialog appears.

Creating New Sources/Sinks

A new instance of a source/sink of a particular type is created by selecting the Add button for the type. This creates a new source/sink and displays a default value or set of values for the source/sink in the edit fields on the right side of the dialog. If the simulation is steady state, normal edit fields are used to enter the values. If the simulation is transient, buttons appear which can be used to bring up the XY Series Editor. A value is entered for each of the defined stress periods.

Multiple instances of a source/sink can be created by repeatedly selecting the Add button. Each cell can contain multiple sources/sinks of a particular type or a mixture of several types.

Editing Sources/Sinks

An existing source/sink can be edited by selecting the name of the source/sink in the source/sink list. The values associated with the highlighted source/sink are displayed in the spread sheet.

Deleting Sources/Sinks

An instance of a source/sink may be deleted by selecting the source sink from the list and selecting the Delete button.

Constant Values

Several of the parameters shown in the Point Sources/Sinks dialog, such as elevation and conductance, are not likely to change during a transient simulation. A constant value can be entered for these parameters in the XY Series Editor and the resulting curve is a flat line.

Areal Sources/Sinks

The Areal Sources/Sinks command is used to edit recharge and evapotranspiration parameters on a cell-by-cell basis. Before selecting the Areal Sources/Sinks command, a set of cells should be selected using the cell selection tools. Recharge and evapotranspiration parameters are applied to vertical columns rather than to individual cells. Therefore, to edit the value for a vertical column, any cell in the column can be selected. Once the Areal Sources/Sinks command is selected, the Areal Sources/Sinks dialog appears. The options in the dialog are as follows:

Packages

If the Recharge package has not been activated (using the Packages dialog which is accessed through the Global Options\Basic Package dialog) the Recharge portion of the Areal Sources/Sinks dialog is dimmed. Likewise, if the Evapotranspiration package has not been activated, the Evapotranspiration portion of the dialog is dimmed.

Editing Values

The edit fields to the right of each option are used to enter the values of the parameters associated with each type. The fields are standard edit fields if the simulation is steady state and are graphic windows displaying a time series if the simulation is transient. Clicking on the window brings up the XY Series Editor. The XY Series Editor is a general purpose editor for entering curves or lists of pairs of data (e.g., rainfall rate vs. time). Once a curve is defined in the editor, it is displayed graphically in the window.

Layers

The Layer option in the Recharge portion of the dialog is only active (undimmed) if the Recharge at Specified Vertical Cells option is chosen in the Recharge Package dialog. Likewise, the Layer option in the Evapotranspiration portion of the dialog is only active if the ET at Specified Vertical Cells option is chosen in the Evapotranspiration Package dialog.

Editing Multiple Cells

After a set of source/sink parameters has been defined, it is often necessary to change one of the parameters of a subset of the cells (vertical columns).

For example, suppose the Evapotranspiration package has been activated and each vertical column was assigned a unique value of elevation, extinction depth, and maximum evapotranspiration rate. Suppose that it becomes necessary to change the extinction depth of a large subset of the vertical columns to a constant value while leaving the other parameters unchanged. The Areal Sources/Sinks dialog has been designed so that it can be used to change one of the parameters without altering the other parameters.

If more than one cell is selected when the Areal Sources/Sinks dialog is brought up, the available edit fields appear with a checkerboard pattern if the values in the cells are not all identical. Before editing a parameter, click on the field to switch it from multi-select mode to normal mode. When the OK button is selected after the parameters have been edited, only the parameters that have been selected are changed. For example, in the case of changing the extinction depth, if the extinction depth is edited, the extinction depth of all of the vertical columns associated with the selected cells is changed to the new value but the individual values of elevation and maximum evapotranspiration rate are left unchanged.

6.3.3. MODFLOW Post-Processing

MODFLOW Display Options

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The properties of all MODFLOW data that GMS displays on the screen can be controlled through the MODFLOW tab of the Display Options dialog. This dialog is opened by right clicking on the MODFLOW entry in the Project Explorer and selecting the Display Options command. It can also be accessed from the from the Display menu, the MODFLOW menu, or the Display Options macro. The following table describes the display options available for the 3D Grid module.

Display Option	Description
Wells, Drains, Rivers, Streams, General head, Constant head, Transient head	These items on the left side of the dialog represent source/sink objects. If the check box just to the left of each source/sink name is selected, a symbol is displayed at the center of each cell with that type of source/sink. The symbol for each source/sink is displayed to the left of the check box. The symbol can be changed by selecting the symbol button. This brings up the symbol editor dialog. The symbol editor contains a list of available symbols. The dialog can also be used to edit the size and color of the symbol.
Horiz. flow barriers	The Horiz. flow barrier option displays a line at the location of each horizontal barrier. The attributes (thickness, color, etc.) can be edited by clicking on the small window to the left of the Horiz. flow barrier toggle.
Dry cells	If the Dry cells option is selected, the chosen symbol will be displayed at the location of all dry cells in the grid. A MODFLOW solution must be imported to GMS prior to displaying dry cells. When a cell goes dry during a MODFLOW simulation, the HDRY value defined in the BCF Package is assigned as the head value for the cell. If an HDRY value is encountered in the active scalar data set when the display is refreshed, the cell is assumed to be dry.
Water table	If the Water table option is selected, the water table defined by a MODFLOW solution is superimposed on the layer geometry when a grid cross section is displayed in orthogonal mode. The water table is defined as the head value in the uppermost active cell. If the water table display is on, all contours (head, concentration, etc.) are clipped so that they lie at or below the water table. This option is only available if the True Layer mode is active.
Mark flooded cells	The Mark flooded cells option is used in conjunction with the True Layer approach to defining layer data. With the True Layer approach, the top elevation is entered for each layer, regardless of the layer type. With many models, the top layer is an unconfined layer and the top elevation represents the ground surface. For unconfined layers, the top elevation array is not read by MODFLOW. Only the bottom elevation array is used in the calculations. MODFLOW assumes that the top layer extends to an infinite height. It is often the case that the computed water table elevation exceeds the elevation of the ground surface. The Mark flooded cells option is used to draw a symbol at the center of all cells where the computed water table elevation is greater than the top elevation of the top layer in the grid.
Lake cells	If the Lake cells option is selected, a symbol is draw in the center of each cell representing a lake. The symbol, the symbol size, and symbol color can be adjusted by clicking on the button to the left of the Lake cells toggle.
Lake ID's	If the Lake ID's option is selected, each cell representing a lake will display the lake ID in the center of the cell. The font, font size, and font color can be adjusted by clicking on the button to the left of the Lake ID's toggle.
Display symbol legend	If the Symbol legend option is selected, a legend showing each of the symbols associated with sources/sinks, dry cells, and flooded cells is displayed in the lower right corner of the GMS window.
Display hydrogeologic units	The Display hydrogeologic units option is used to display HUF data. This option only works if the current display mode is Ortho Mode. This mode can be turned on in the Display menu.
Check All	By clicking this button, all of the display options are turned on.
Check None	By clicking this button, all of the display options are turned off.

MODFLOW Post Processing Viewing Options

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

In addition to generating and editing the MODFLOW input files, GMS can also be used for post-processing the solution files computed by MODFLOW. The solution files computed by MODFLOW include head, drawdown, and CCF files. By default, the head and CCF files are automatically generated. The output options are controlled in the MODFLOW *Output Control* dialog.

No-Flow and Dry Cells

When a MODFLOW simulation is solved, MODFLOW writes out a head or drawdown value for every cell of the finite difference grid to the solution files. However, some of the cells are either outside the problem domain or they have gone dry during the course of the simulation. These cells are flagged by MODFLOW in the output file by writing special values for the cells. The value assigned to inactive cells is the No flow head value specified in the *Global Options/Basic Package* dialog. The value assigned to cells which have gone dry is the Head assigned to dry cells value defined in the BCF, LPF or HUF package dialogs. If the MODFLOW data are in memory when the solution is read in, GMS will automatically use the No flow head and Head assigned to dry cells values to define active/inactive cells for post-processing.

Layer Contours

In most cases, the best way to display computed head and drawdown is with layer contours. Layer contours are generated by selecting the Layer Contours option in the 3D Grid Display Options dialog. This option is automatically turned on whenever a MODFLOW solution is read into GMS. The **Mini-Grid Toolbar**, which is part of the Tool Palettes, can be used to switch between layers.

Viewing Computed Fluxes

The CCF file that is part of the MODFLOW solution contains useful information about the computed flux rates between the aquifer and external sources and sinks. A special set of tools are provided for viewing computed fluxes.

Summation of Fluxes on Arcs and Polygons

GMS writes the necessary files so that MODFLOW will output the computed flow for boundary conditions created in the Map module. These computed flows can be viewed by selecting the appropriate feature object (river arc, etc) and then looking at the Status Bar. For an object with an assigned observed flow, the observed flow value and the residual error are displayed in addition to the computed flow. For objects without an observed flow, only the computed flow is shown.

This cannot be done for SFR stream arcs since the SFR package does not participate in the Observation Process.

Calibration Targets and Statistics

For objects with an observed flow, a Calibration Target can be plotted on the object. The calibration target provides a graphical representation of the calibration error. Calibration targets are described in section 14.5. The display of flow calibration targets is turned on by selecting the Display Options command in the Feature Objects menu when the Local Source/sink coverage is the active coverage.

Flow Budget for Selected Cells

If a MODFLOW model is built without using the conceptual model approach (directly from the grid), the computed fluxes corresponding to a user-defined set of cells can still be displayed. This is accomplished by selecting a set of cells and selecting the ***Flow Budget*** command from the *Data* menu in the 3D Grid module. This command brings up the *Flow Budget* dialog. This dialog lists the complete flow budget for the selected cells.

GMS also includes a **Zone Budget** tool to view the computed flows into (out of) user defined zones.

Vector Plots

If a CCF file has been imported as described above, a vector plot can be generated to illustrate the flow field computed by MODFLOW. The CCF file contains flows through each of the cell walls in the grid, i.e., the flow from each cell to each of its six surrounding cells.

Vectors are generated by right-clicking on the CCF file in the Project Explorer and selecting the ***CCF->Velocity Vectors*** command. The cell top and bottom elevations as well as the porosity are used when calculating the velocity vectors. Therefore, for GMS to compute the vectors, a MODFLOW simulation must exist in GMS. Although porosity is not an input to a MODFLOW model, the porosity can be specified per cell, using the cell properties dialog, whenever a MODFLOW simulation exists.

The vectors are computed by reading the CCF file to get the net flow through each grid cell face. Each component of the velocity vector is computed by dividing the flow through the cell face by the saturated area of the face multiplied by the porosity. For example, the I, J, K direction components of the vector are calculated as follows:

- $vector_i = \frac{flow_i}{(saturated_{Area_i})(porosity)}$
- $vector_j = \frac{flow_j}{(saturated_{Area_j})(porosity)}$
- $vector_k = \frac{flow_k}{(area_k)(porosity)}$

The saturated area may be less than the area of the cell face if the computed head is below the top elevation of the grid cell. The velocity vector is set to 0.0 in cells that are inactive, dry (the computed head is below the cell bottom elevation), or have inappropriate elevations (the top elevation is below the bottom elevation).

Prior to version 8.2.

To generate a vector data set from the CCF file, right click on the CCF data set in the Data Tree and select the **Generate vectors** option from the pop up menu. A flow vector is generated at each cell center by computing a vector sum of the flows through the six walls of the cell. The resulting vectors can be plotted by selecting the Vectors option in the *3D Grid Display* Options dialog.

Viewing the Printed Output File

Two types of output are produced by MODFLOW: a printed output file and a set of solution files (head, drawdown, CCF). Before reading in the solution files, it is often useful to examine the printed output file. In some cases, MODFLOW may crash or not complete its run successfully. You can usually determine if the run was completed successfully by viewing the printed output file. When viewing the file you should check to make sure that a solution was output for all stress periods and time steps you are expecting. In some cases MODFLOW will also output to the listing file a description of any problems which may have occurred.

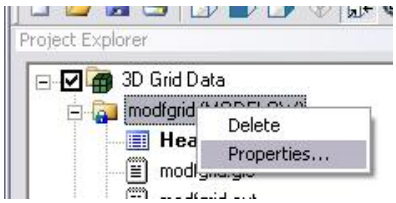
GMS provides two ways to view the text files produced by MODFLOW and the other analysis codes:

1. When a solution is read into GMS, the text output files are placed into the data tree. Double clicking on their file icons in the Project Explorer will bring up the text files in a text editor.
2. Any text file can be viewed by selecting the ***Edit File*** command in the *File* menu. A File Browser appears and the selected file is opened in a text editor.

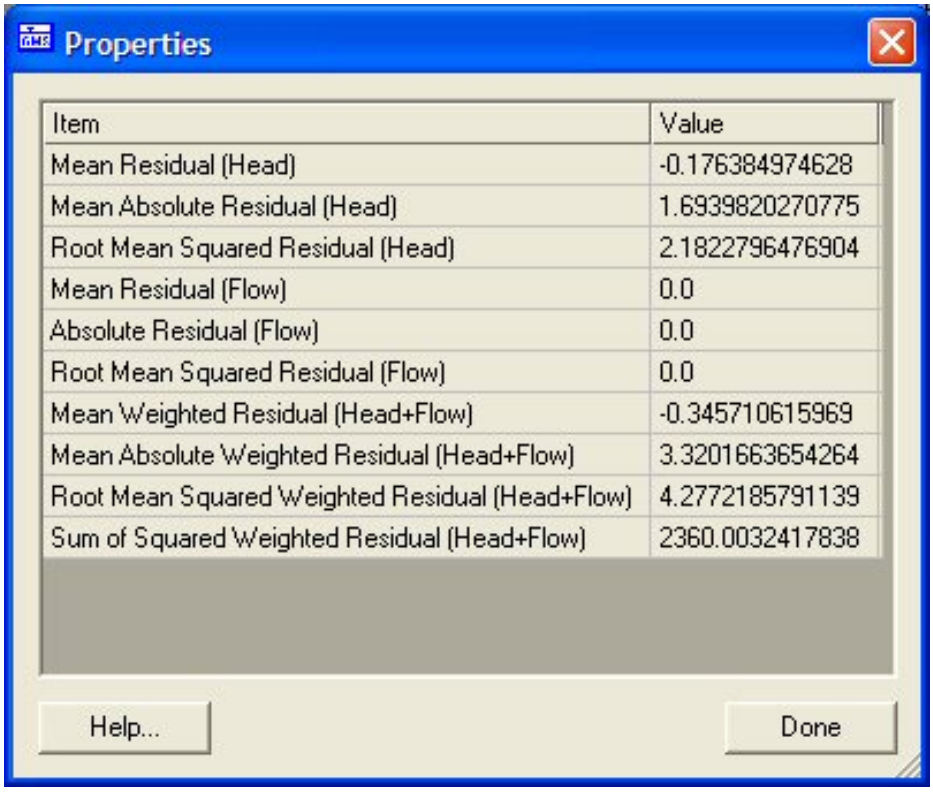
MODFLOW Solution Properties Dialog

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The MODFLOW Properties Dialog Box is opened by right clicking on the MODFLOW solution folder in the project explorer.



The data in this dialog come from the *._os, *._ww, *._r, *._w files computed by the MODFLOW Observation process. The computation from MODFLOW will include any observed flows combined with the observed heads to calculate a single error value. In a transient model, the error value includes all of the observations at the various times. The error shown in this dialog is different from the error found in the Error Summary Plot.



Comment on different error values

- Mean Residual - Average error for the observations. This can be misleading because the positive and negative errors can cancel.
- Mean Absolute Residual - Mean of the absolute error values for the observations. This is a true mean, not allowing positive and negative errors to cancel.
- Root Mean Squared Residual - RMS is calculated by taking the sum of the square of the errors for the observations and then taking its square root. This tends to give more weight to cases where a few extreme error values exist.
- Sum of Squared Weighted Residual - This is the error value that is minimized by a PEST run.

Point head observation errors

Mean Residual (Head).

Mean Absolute Residual (Head).

Root Mean Squared Residual (Head).

Flow observation errors

Mean Residual (Flow).

Mean Absolute Residual (Flow).

Root Mean Squared Residual (Flow).

Combined head and flow error values

Mean Weighted Residual (Head+Flow).

Mean Absolute Weighted Residual (Head+Flow).

Root Mean Squared Weighted Residual (Head+Flow).

Sum of Squared Weighted Residual (Head+Flow).

Saving and Reading a MODFLOW Simulation

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Saving a MODFLOW Simulation

Once a MODFLOW simulation has been created and checked for potential problems with the Model Checker, the next step is to save the simulation to disk and run MODFLOW. MODFLOW simulations are saved using the ***Save*** and ***Save As*** commands in the *File* menu.

Selecting the ***Save*** command saves the GMS project including the model simulation. By default the model simulation will be saved to the same location as the GMS project. However, in the Save dialog the path for the model simulation can be specified.

A MODFLOW simulation is actually saved to a set of input files. The MODFLOW super file is used to store extra data used by GMS but not used by MODFLOW. The super file also contains the path to the name file. The names of all of the input and output files associated with a simulation are saved in the name file. When MODFLOW is launched, the name of the super file is automatically passed to the MODFLOW executable.

When a MODFLOW simulation is saved, the names of the other MODFLOW input files are automatically patterned after the name of the super file. For example, if the super file is named sampmod.mfs, the other files are named sampmod.bas, sampmod.bcf, etc.

The user can adjust the output options in the *Output Control* dialog

The version of MODFLOW 2000 that ships with the GMS software has been modified to be able to read and write input data stored in HDF5 files. For more information see MODFLOW with HDF5.

Reading a MODFLOW Simulation

Once a MODFLOW simulation has been saved by GMS using the ***Save*** command in *File* menu, the entire simulation can be read back into GMS using the ***Open*** command in the *File* menu.

MODFLOW simulations can be read in individually (*.mfs) or as part of a project (*.gpr). They can also be imported by using the MODFLOW name file. The extension for the name file must be changed to *.mfn.

Reading a Completed MODFLOW Solution

When you are using the Model Wrapper to control your MODFLOW run, you can use the **Read MODFLOW solution** toggle at the bottom of the dialog to automatically read in the corresponding solution.

You can also read in solution files generated by MODFLOW using the ***Read Solution*** command in the *MODFLOW* menu. This command brings up the a file browser and prompts you to choose a MODFLOW super file (pre GMS 7.0) or a MODFLOW name file (GMS 7.0+).

Cell Properties

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

The cell properties dialog allows the user to edit cell properties. Most cell properties are associated with a model such as MODFLOW or MT3D. If no models exist in the GMS project them the cell properties dialog will only allow editing of the material assigned to the grid cell.

MODFLOW

Several input arrays defining parameters such as starting head, IBOUND, hydraulic conductivity, and transmissivity are defined in the Global/Basic and BCF, LPF, or HUF packages. These arrays can be edited in the Basic and BCF/LPF/HUF Package dialogs, or they can be initialized using a conceptual model in the Map module. In many cases however, it is necessary to view or edit the values on a cell-by-cell basis. This can be accomplished using the ***Cell Properties*** command in the *MODFLOW* menu.

Before selecting the ***Cell Properties*** command, a set of cells should be selected using the cell selection tools. Once the command is selected, the *MODFLOW Cell Attributes* dialog appears.

The parameters for the selected cells are changed by typing in new values in the edit fields. If more than one cell is selected when the ***Cell Properties*** command is selected, the available edit fields will be left blank (unless all values are the same for that parameter). To edit one of the parameters, click on the edit field to be edited. When the **OK**

button is selected, only the parameters whose edit fields that have data are changed. This makes it possible to change one of the parameters (e.g., transmissivity) for all of the selected cells while leaving the other parameters unchanged.

NOTE: When you are using materials to define the MODFLOW model, the Cell Properties dialog will show the material properties relating to the material of the selected cell. You will not be able to edit these values on a cell-by-cell basis, but you can either edit the material type for this cell if the active material set is the default material set, or you can change the material properties for the material (which affects every cell that uses that material).

MT3DMS/RT3D/SEAM3D

MT3D inputs that vary on a cell by cell basis can also be editing using this dialog and editing the data in the MT3D tab.

MODFLOW World File

A MODFLOW world file is a plain text file containing the geographic location of a MODFLOW model. The world file contains 3 lines including the origin, rotation about the z axis, and the path of a projection file. The world file is found in the same directory as the name file. The MODFLOW world and projection files are automatically written by GMS when the project has a global projection.

Format

```
ORIG x y z
ROTZ theta
PRJ_FILE "file_path"
```

The ORIG line contains the space separated x, y, and z coordinates of the outside corner of the model cell at the first column and last row.

The ROTZ line contains the counter-clockwise (looking down) rotation of the model about the origin coordinate in degrees.

The PRJ_FILE line contains the absolute or relative path to the model's projection file which ends in with a *.prj extension. The file path is sourounded by double quotes.

Sample

```
ORIG 612086.0 3429376.55 230.0
ROTZ 90.0
PRJ_FILE "modfmap.prj"
```

Error Summary Plot

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

An Error Summary plot is one of the plot types that can be created by the Plot Wizard. See the Error Summary plot section on the Plot Wizard page for a basic description. The Error Summary plot is used to display a text listing of the mean error, mean absolute error, and root mean squared error for an observation measurement in a coverage. The error values may be listed for a single time step of a data set or for all time steps.

Error types

Mean Error: This is the average error for the points. This value can be misleading since positive and negative errors can cancel.

Mean Absolute Average: This is the mean of the absolute values of the errors. It is a true mean, not allowing positive and negative errors to cancel.

Root Mean Square: This takes the sum of the square of the errors and then takes its square root. This norm tends to give more weight to cases where a few extreme error values exist.

Time matching

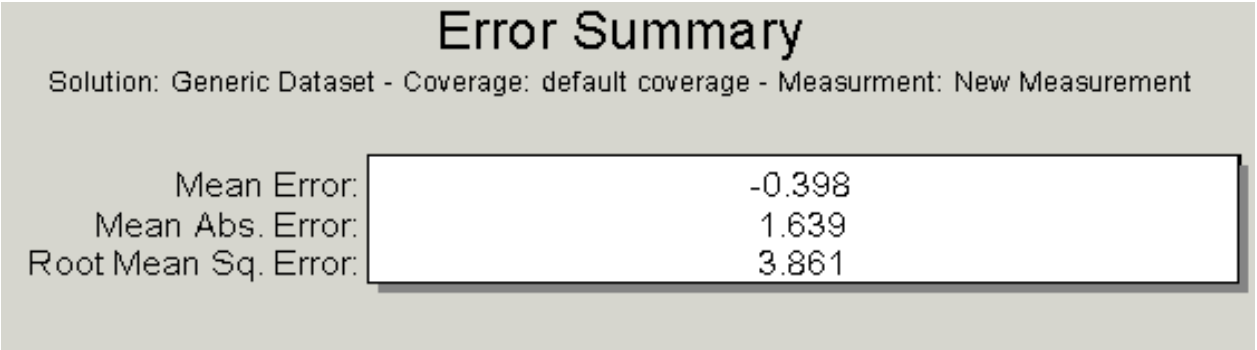
When the measurement is transient (i.e., observed heads at various times) then it is very unlikely that the output times from the model match the times when the field measurements were taken. In this case GMS will interpolate an observed value at the model output time. If the model output time is before the first field measurement for a particular point then the interpolated value is the first measurement. Similarly if the model output time is after the last field measurement then the interpolated value is the last measurement. When the model output time is between the first and last measurement then GMS will find the 2 closest field measurments to the model output time and use linear interpolation to compute the "observed" value at that time.

MODFLOW errors

The Error Summary Plot is most useful with models that do not make error computations from observation data. MODFLOW does compute error computations from observation data, and these errors are available in the MODFLOW properties dialog. Therefore, when using MODFLOW, you should use the MODFLOW properties dialog and not the Error Summary Plot. The errors shown in the Error Summary Plot are not the same as those found in the MODFLOW properties dialog due to the time matching and interpolation issues described above. The MODFLOW properties dialog shows the errors computed by MODFLOW for the observations at the actual observation times and is therefore superior. The errors reported by MODFLOW have nothing to do with the model output times, only observation times.

Plot creation

Error Summary plots are created in the Plot Wizard by setting the plot type to Error Summary. A sample plot is shown in the figure.



Error Summary Plot Options

After the plot type is set in the first step of the Plot Wizard, the Next button is clicked to move to the second step of the Plot Wizard.

Coverage: Displays the name of the coverage where the current data for the plot is coming from.

Measurement: This is the name of the current measurement, created in the Feature Objects | Attributes dialog, being plotted.

Feature Objects: Displays which feature object is utilized in the current plot, points or arcs.

Zone Budget

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

In previous versions of GMS, it has been possible to view a detailed MODFLOW flow budget for a set of selected cells using the Flow Budget command in the Data menu. Now, the flow budget can be summarized based on user-defined zones. This provides a capability quite similar to the USGS ZoneBudget tool. However, the calculations are all done inside GMS using the CCF file generated by MODFLOW.

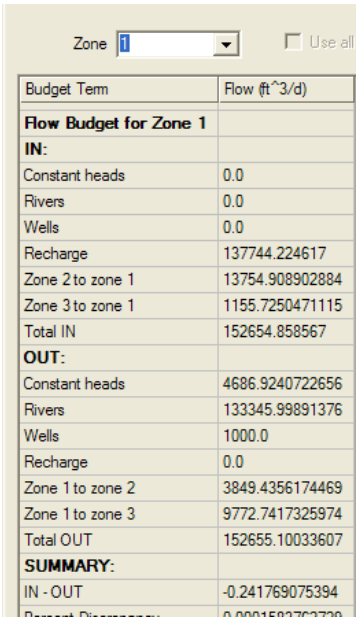
Defining Zones

The first step in using this utility is to identify the model zones by assigning a zone budget id to each of the cells in the grid. The zone budget id is an integer value. It can be associated with polygons in a MODFLOW conceptual model in the map module. The zone budget id property is listed in the Areal Properties column of the Coverage Setup dialog. These IDs are then assigned to the appropriate grid cells when the ***Map->MODFLOW*** command is selected. The IDs can also be assigned directly to cells by selecting a set of cells and selecting the Cell Properties command in the MODFLOW menu.

A Zone Budget IDs item is also included just below the MODFLOW icon in the Project Explorer window. Clicking on this icon displays the zone budget IDs and double-clicking brings up the zone budget id array in the spreadsheet editor.

Viewing the Flow Budget

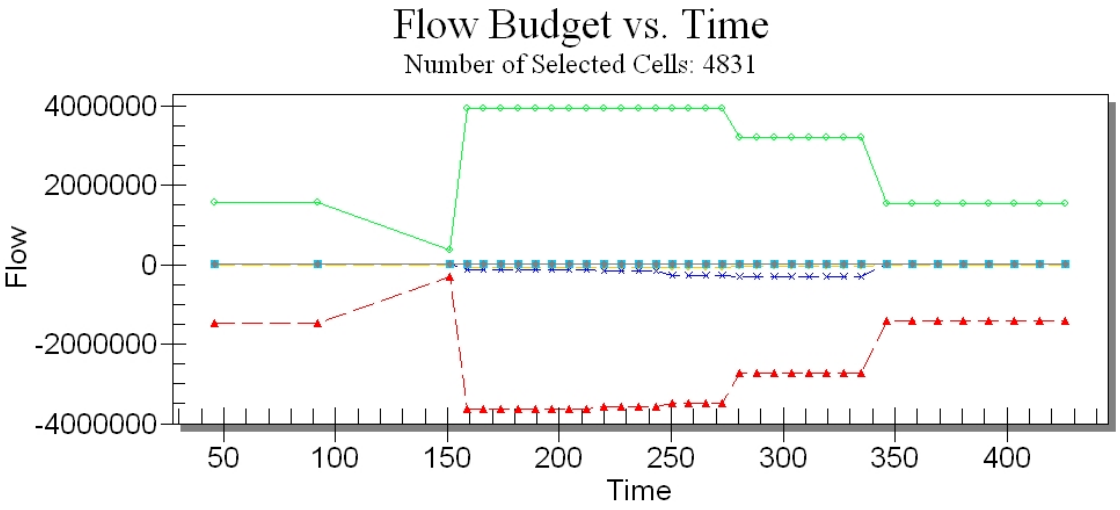
Once the zone budget IDs have been assigned, the flow budget for each zone can be viewed by selecting the Flow Budget command in the MODFLOW menu. This brings up the Flow Budget dialog. There is a new zone selector combo box at the top of the dialog. The flow budget associated with the selected zone is shown in the table. The combo box includes an "All zones" option that displays a summary for all zones in the model. Also, turning on the **Use all timesteps toggle** will create a flow budget report for each time step in the current MODFLOW solution.



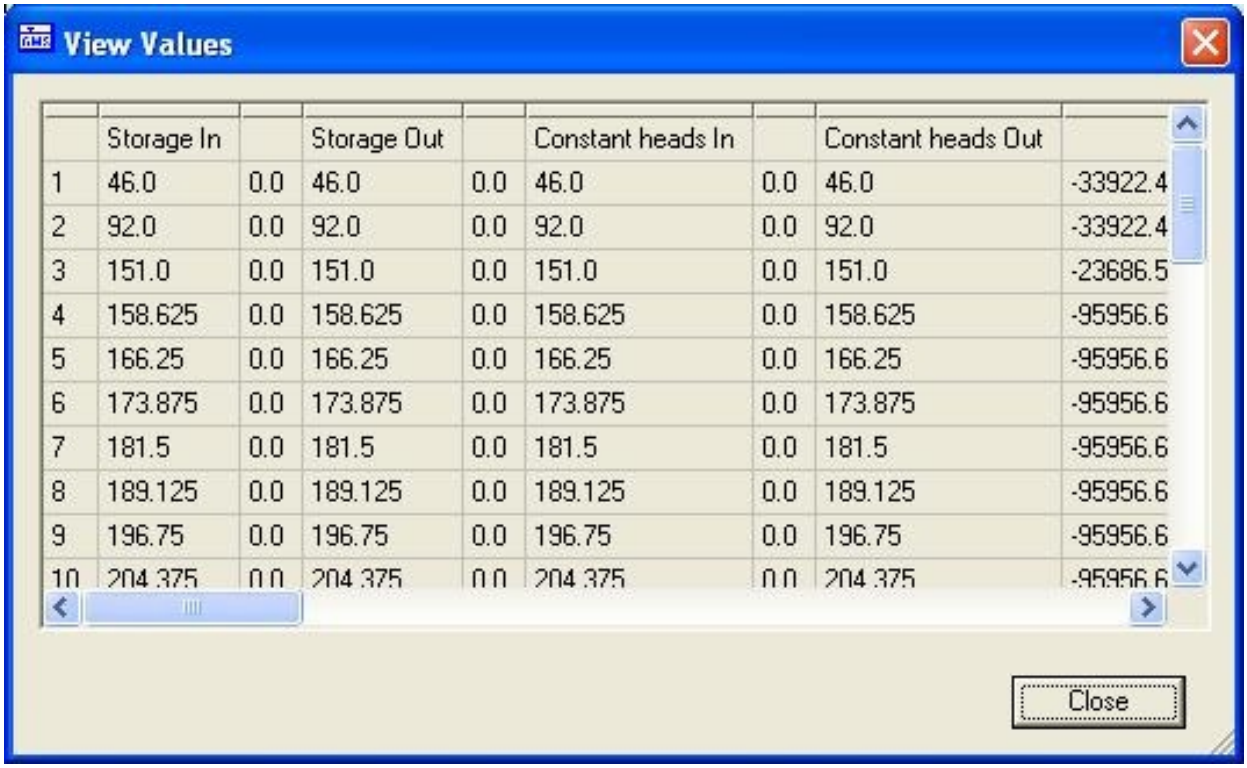
Budget Term	Flow (ft^3/d)
Flow Budget for Zone 1	
IN:	
Constant heads	0.0
Rivers	0.0
Wells	0.0
Recharge	137744.224617
Zone 2 to zone 1	13754.908902884
Zone 3 to zone 1	1155.7250471115
Total IN	152654.858567
OUT:	
Constant heads	4686.9240722656
Rivers	133345.99891376
Wells	1000.0
Recharge	0.0
Zone 1 to zone 2	3849.4356174469
Zone 1 to zone 3	9772.7417325974
Total OUT	152655.10033607
SUMMARY:	
IN - OUT	-0.241769075394

Viewing in a Plot File

For a transient model you may use the flow budget vs. time plot to view the flow budget for any number of cells.



By right clicking on the plot and select the view values option, GMS will generate a table of the flow budget for the different time steps.

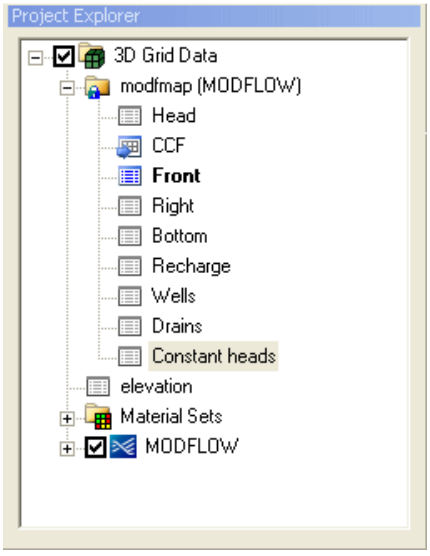


	Storage In		Storage Out		Constant heads In		Constant heads Out	
1	46.0	0.0	46.0	0.0	46.0	0.0	46.0	-33922.4
2	92.0	0.0	92.0	0.0	92.0	0.0	92.0	-33922.4
3	151.0	0.0	151.0	0.0	151.0	0.0	151.0	-23686.5
4	158.625	0.0	158.625	0.0	158.625	0.0	158.625	-95956.6
5	166.25	0.0	166.25	0.0	166.25	0.0	166.25	-95956.6
6	173.875	0.0	173.875	0.0	173.875	0.0	173.875	-95956.6
7	181.5	0.0	181.5	0.0	181.5	0.0	181.5	-95956.6
8	189.125	0.0	189.125	0.0	189.125	0.0	189.125	-95956.6
9	196.75	0.0	196.75	0.0	196.75	0.0	196.75	-95956.6
10	204.375	0.0	204.375	0.0	204.375	0.0	204.375	-95956.6

Viewing the MODFLOW CCF File

There are two new options to view MODFLOW cell-to-cell flow (CCF) file output:

1. The CCF file can be viewed in a spreadsheet by right-clicking on the CCF file in the Project Explorer and selecting the View Values command.
2. The CCF file can also be converted to a set of data sets. One data set would be created for drains (if they exist), one for rivers etc., and 3 for cell-to-cell flow (flow right, flow back, flow down).



CCF data sets displayed in the Project Explorer

ZONEBUDGET

ZONEBUDGET	
Model Info	
Model type	Analysis of MODFLOW cell to cell flow files
Developer	USGS, [1]
Documentation	ZONEBUDGET 1.0 [2] ZONEBUDGET 3.01 [3]
Tutorials	MODFLOW Tutorials

ZONEBUDGET is USGS program that reads cell to cell flow data produced by MODFLOW and calculates water budgets for subregions of the modeled area. GMS has an interface to ZONEBUDGET available from the ***MODFLOW | Flow Budget*** menu command. This command brings up the Flow Budget dialog and one of the available tabs in the dialog is USGS ZONEBUDGET. GMS supports version 3.01 of ZONEBUDGET.

When the user brings up the ZONEBUDGET tab they will be analyzing the CCF file from the active MODFLOW solution. All ZONEBUDGET inputs and outputs will be written to the same location as the CCF file.

GMS has its own internal tools for analyzing CCF data. These tools are explained here.

ZONEBUDGET OPTIONS

- **Title** - This is a title that is printed to the listing output file.
- **Output**
  - **Listing File** - This specifies that ZONEBUDGET will produce a listing file as output. This was the only kind of output that the first version of ZONEBUDGET produced.
  - **Multiple Outputs** - Version 3 of ZONEBUDGET has the option to produce multiple output files.
    - **ZBLST** - ZONEBUDGET will create a listing file as part of its output.
    - **CSV** - ZONEBUDGET will create a comma separate value (CSV) file as part of its output. For each time for which a budget is requested there is one line for each input term and each output term along with totals. The zones are displayed in columns, so one table displays all zones. This makes it easy to compare any budget term for all the different zones.
    - **CSV2** - ZONEBUDGET will create another CSV file as part of its output. The CSV2 file displays the complete budget for one zone and one time in a single line. Each column has a separate inflow or outflow budget term. The rows can be sorted by time within the spreadsheet program, which makes it possible to easily see how each term changes with time.
  - **Output times**
    - **all** - Every time step in the MODFLOW CCF file will be processed.
    - **specified** - With this option selected the user can select individual time steps from the CCF file for ZONEBUDGET to process. When the user clicks on the **Times** button a spread sheet comes up that allows the user to select the time steps.
- **Composite Zones** - ZONEBUDGET allows users to combine zones into "Composite Zones" and then information about the composite zone is printed to the listing file.
  - **Number of zones** - This specifies the number of composite zones.
  - **Edit Zones** - This button brings up a dialog where the user can create composite zones. A composite zone is defined by a name that is 10 characters or less in length and a list of the zone ids separated by a comma. For

example, when analyzing a model with zones 1, 2, 3, and 4, composites zone could be defined as follows:

Name	Zone IDs
zone_top	1, 2
zone_bot	3, 4

- **Run ZONEBUDGET** - This launches ZONEBUDGET with the specified input options.
- **View** - This allows the user to load ZONEBUDGET outputs. The options available include LST, ZBLST, CSV, and CSV2. The contents of the output file are loaded into a text window or a spread sheet in the space below the View check box.
- **Open Containing Folder** - This button brings up Windows Explorer opened to the directory with the ZONEBUDGET output.

References

[1] http://water.usgs.gov/nrp/gwsoftware/zonebud3/zonebudget3.html
[2] http://gmsdocs.aquaveo.com/ofr90392.pdf
[3] http://gmsdocs.aquaveo.com/zonbud3.pdf

Viewing Computed Fluxes

Once a set of observed flows has been entered, the next step is to run the MODFLOW model, read in the solution, and compare the computed flows to the observed flows. The computed flows are part of the MODFLOW output.

Summation of Fluxes on Arcs and Polygons

GMS writes the necessary files so that MODFLOW will output the computed flow for boundary conditions created in the Map module. These computed flows can be viewed by selecting the appropriate feature object (river arc, etc) and then looking at the Status Bar. For an object with an assigned observed flow, the observed flow value and the residual error are displayed in addition to the computed flow. For objects without an observed flow, only the computed flow is shown.

Calibration Targets and Statistics

For objects with an observed flow, a calibration target can be plotted on the object. The calibration target provides a graphical representation of the calibration error. Calibration targets are described in section 14.5. The display of flow calibration targets is turned on by selecting the ***Display Options*** command in the *Feature Objects* menu when the Local Source/sink coverage is the active coverage.

Flow Budget for Selected Cells

If a MODFLOW model is built without using the conceptual model approach (directly from the grid), the computed fluxes corresponding to a user-defined set of cells can still be displayed. This is accomplished by selecting a set of cells and selecting the ***Flow Budget*** command from the *Data* menu in the 3D Grid module. This command brings up the *Flow Budget* dialog. This dialog lists the complete flow budget for the selected cells.

6.4. MODPATH

MODPATH

MODPATH	
Model Info	
Model type	particle tracking
Developer	USGS, [1]
Documentation	MODPATH Manual [1]
Tutorials	MODPATH Tutorials

MODPATH is a particle tracking code that is used in conjunction with MODFLOW. After running a MODFLOW simulation, the user can designate the location of a set of particles. The particles are then tracked through time assuming they are transported by advection using the flow field computed by MODFLOW. Particles can be tracked either forward in time or backward in time. Particle tracking analyses are particularly useful for delineating capture zones or areas of influence for wells.

A complete description of MODPATH is beyond the scope of this help file, however, because of the user-friendly interface in GMS, an intimate knowledge of MODPATH is not required to effectively do particle tracking in GMS. For a more complete knowledge of MODPATH, refer to the MODPATH documentation (Pollock, 1994).

MODPATH was developed by the U.S. Geological Survey. Version 5.0 of MODPATH is supported in GMS. The version of MODPATH distributed with GMS is the original public domain version distributed by the USGS, with minor modifications to accommodate GMS.

MODPATH Links

- MODPATH Particle Tracking
- MODPATH Particle Sets
- MODPATH Zone Codes
- MODPATH Display Options
- Exporting Pathlines

References

[1] http://gmsdocs.aquaveo.com/mpathref.pdf

MODPATH Particle Tracking

Requirements

Before you can do particle tracking with MODPATH, you must have:

1. A MODFLOW simulation in memory.
2. A MODFLOW solution in memory computed using MODFLOW 2000 with a valid head and flow field.

If the Conceptual Model approach is used to build the MODFLOW model, the same conceptual model can be used to initialize some of the input data for MODPATH. Both the zone codes and porosities can be defined using polygons in a MODFLOW / MT3D / MODPATH Layer Attributes type coverage. The values are assigned to the cells when the ***Map -> MODPATH*** command is selected.

Steps

Once you have a MODFLOW solution, all that is necessary to do particle tracking is to:

1. Create particle starting locations.

As soon as you create the particle starting locations GMS immediately does a number of things depending on the *Run MODPATH automatically for steady state models* option in the *MODPATH General Options* dialog. If the option is on and the MODFLOW model is steady state.

Automatic option on

1. GMS saves the MODPATH input files.
2. GMS launches MODPATH and passes it the input files.
3. GMS waits for MODPATH to compute the pathlines.
4. GMS then reads in and displays the pathlines.

All this is done automatically and behind the scenes and usually takes just a second or two. Thus, pathlines will appear almost immediately after you define the particle starting locations. If MODPATH encountered problems attempting to generate the path lines, the errors will be displayed in the MODPATH Errors dialog.

Automatic option off

If the *Run MODPATH automatically for steady state models* option is off, or the MODFLOW model is transient, you must save and run MODPATH manually after creating particle starting locations before pathlines will appear. This is to prevent having GMS spend lots of time updating pathlines because MODPATH can take a considerable amount of time with large, transient models. The *MODPATH|Run MODPATH* menu command or equivalent toolbar macro can be used to run MODPATH.

Generating Particles

There are two ways to generate particle starting locations in GMS, both available via the MODPATH menu:

Generate Particles at Wells - Accessed in the *MODPATH* menu. This command automatically generates particle starting locations in cells containing wells. The specified number of particle starting locations are distributed evenly in a horizontal circle around the center of the cells containing the wells. You specify the number of starting locations per cell, and whether you want starting locations created in extraction well cells, or injection well cells.

Injection Wells - If this option is selected, particles are created in cells with wells that have a positive flow rate. The tracking direction for the current particle set is changed to forward.

Extraction Wells - If this option is selected, particles are created in cells with wells that have a negative flow rate. The tracking direction for the current particle set is changed to backward.

Selected Wells - This option makes it possible to generate particles at either all wells or only at wells within selected cells.

Generate Particles at Selected Cells - Accessed in the *MODPATH* menu. This command opens the *Generate Particles* dialog, and when you select **OK**, particles are created in all the selected cells according to the options specified.

Number of Particles - If the **More options** toggle is not selected, you can use the slider to change the number of particles created per cell. The minimum is 1, and the maximum depends on the option being used to distribute the particles. The number of particles jumps as the slider is moved in order to create an even distribution of particles. If the **More options** toggle is selected, the slider is not available, and the user has more control over the total number of particles and how they are distributed.

Distribute Particles - There are three options for distributing the particles within the selected cells.

- **Within Cell** - With this option, the particles are distributed in the interior of the cells.
- **On Cell Faces** - With this option, the particles are distributed on the faces of the cells.
- **Distribute Starting Points on Water Table Surface** - With this option, the particles are restricted to the top face only. When using this option, you should be sure to select the cells corresponding to the location of the water table surface.
- **More Options** - The More options toggle expands the dialog, allowing you to further define how particles are distributed within the selected cells.

(These commands are only available if a MODFLOW CCF solution generated by MODFLOW 2000 exists, and you have 3D grid cells selected.)

Transient MODFLOW Solution

For each time step, particle paths are computed just as for the steady state case until the end of the time step is reached. A new velocity distribution is then calculated for the next time step and the computation of particle paths is resumed. The computation of paths forward or backward, boundary conditions, and the path line termination criteria are handled the same as for steady state flow.

MODPATH Particle Sets

MODPATH particles are grouped into sets. For each set, you can specify the name of the set, the tracking direction, the tracking duration, and if the particles in the set should be displayed.

Grouping particles into sets increases the power and flexibility of particle tracking. For example, several particle sets, some tracking backward and some forward can all be displayed at once. Or, several particle sets, each with different tracking times, can all be displayed at the same time. This can be used to show nested capture zones.

Active Particle Set

One particle set is always designated as the active particle set, and there is never more than one active particle set. Whenever particles are created, they are added to the active particle set. Particles can only be deleted from the active particle set - to delete particles in a particle set that is not active, you must first make it active.

Order

The order that particle sets are listed in the *Data Tree* corresponds to the order they are displayed. Thus, particle sets that are listed at the top are displayed on top of particle sets listed further down. The order can be changed by dragging the particle sets.

Creation / Deletion

If a particle set is deleted, all the particles in the set are deleted.

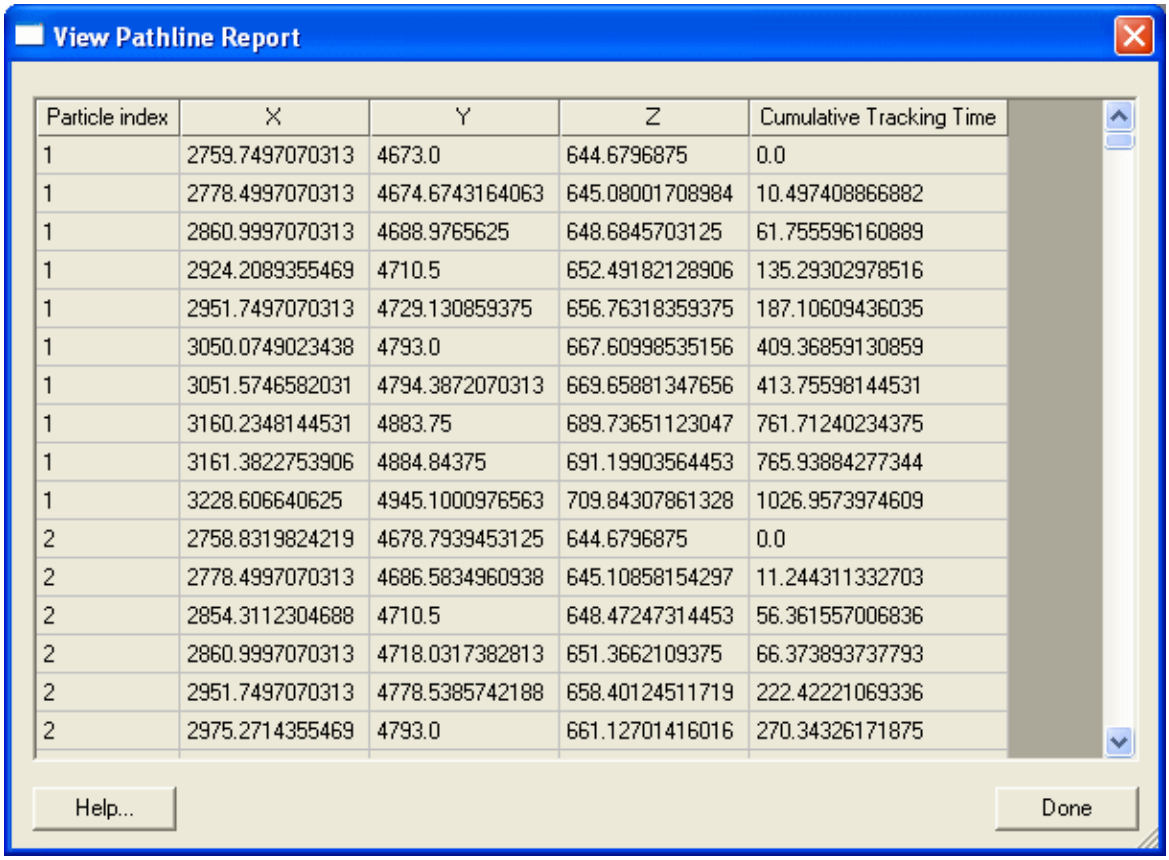
If there are no particle sets, and you create particles, a particle set is automatically created.

Pathline Export

See Exporting Pathlines.

Spreadsheet Report

In addition to the text file export, we have an option for generating a tabular report directly within GMS. This report is accessed by right clicking on a particle set in the Project Explorer and selecting the ***View Pathline Report*** command. This command brings up the following dialog:



Particle index	X	Y	Z	Cumulative Tracking Time
1	2759.7497070313	4673.0	644.6796875	0.0
1	2778.4997070313	4674.6743164063	645.08001708984	10.497408866882
1	2860.9997070313	4688.9765625	648.6845703125	61.755596160889
1	2924.2089355469	4710.5	652.49182128906	135.29302978516
1	2951.7497070313	4729.130859375	656.76318359375	187.10609436035
1	3050.0749023438	4793.0	667.60998535156	409.36859130859
1	3051.5746582031	4794.3872070313	669.65881347656	413.75598144531
1	3160.2348144531	4883.75	689.73651123047	761.71240234375
1	3161.3822753906	4884.84375	691.19903564453	765.93884277344
1	3228.606640625	4945.1000976563	709.84307861328	1026.9573974609
2	2758.8319824219	4678.7939453125	644.6796875	0.0
2	2778.4997070313	4686.5834960938	645.10858154297	11.244311332703
2	2854.3112304688	4710.5	648.47247314453	56.361557006836
2	2860.9997070313	4718.0317382813	651.3662109375	66.373893737793
2	2951.7497070313	4778.5385742188	658.40124511719	222.42221069336
2	2975.2714355469	4793.0	661.12701416016	270.34326171875

MODPATH Zone Codes

When post-processing the results of the MODPATH simulation, the colors of the paths or particles can be varied depending on the zone code of the cell in which the particle started or in which the particle terminated. Zone codes are assigned using the IBOUND array in the MODFLOW interface. In other words, unique values of the IBOUND array (100, 200, 300, etc.) are assigned to the cells of the grid to define the zone codes. The zone code array can be edited by clicking on the ***Zone Code Array*** command in the *MODPATH* menu. This command is only available if a MODFLOW simulation exists.

Cell Properties

The Zone Code for each cell can also be edited by selecting a cell and then choosing the ***Cell Properties*** command from the *MODPATH* menu. In addition to the zone code, cell porosity can be edited in the *Cell Properties* dialog.

MODPATH Display Options

A set of display options unique to the MODPATH input data and MODPATH solutions is provided in GMS. These options are accessed through the ***Display Options*** command in the *MODPATH* menu. This command is only available if a MODFLOW simulation exists. This command brings up the Particles tab of the *3D Grid Display Options* dialog. The following table describes the display options available for the MODPATH model.

Display Option	Description
Starting locations	The starting locations for the particles can be turned on and off. The symbol style, color and size can also be adjusted.
Starting location indices	The starting locations indices for the particles can be turned on and off. The font style, color and size can also be adjusted.
Ending locations	The ending locations for the particles can be turned on and off. The symbol style, color and size can also be adjusted.
Pathlines	The pathlines can be toggled on and off, and the pathline style and color can be adjusted. The pathline color can be set to the default color, or the color corresponding to the starting, ending, or cell zone codes of the pathlines. The "Cell zone code" option causes the pathline to change color as it passes through cells with different zone codes.
Direction arrows	Direction arrows can be displayed along the pathlines. The size of the arrow heads can be set by specifying the Length in pixels. The arrows are displayed along the pathline at an interval corresponding to the Length specified.
Zone codes	Colors can be associated with zone codes. If the Auto compute colors option is selected, GMS will automatically pick colors to go with each unique zone code. Otherwise, you can specify the color you want to associate with each zone code. These colors are used if the Pathline Color option is set to Starting zone code or Ending zone code.
Capture zones	GMS groups pathlines together into capture zones. A polygon representing the outer boundary of the pathlines is found and can be outlined, or filled. The following options are available relating to the capture zones: • **Delineate by well** With this option, all the pathlines originating from a cell containing a well will be grouped together into one capture zone. All wells with the same name are included in the same capture zone. • **Delineate by zone code** With this option, all pathlines originating from the same zone code are grouped together into capture zones. This option could be used to create a single capture zone for several wells that have different names. The cells containing the wells would need to be assigned the same zone code. This option could also be used to find the "capture zone" for a zone of cells where the cells all have the same zone code. • **Thin triangle ratio** GMS triangulates the pathlines in a capture zone to determine the capture zone boundary. Depending on the length and shape of the pathlines, it may be necessary to adjust the Thin triangle ratio to get a good looking capture zone. Increasing the value causes more area around the edges to be included. Decreasing the value causes less area around the edges to be included.

Exporting Pathlines

MODPATH pathlines can be exported from GMS in a few different ways:

Capture Zones -> Arcs

The *MODPATH | Capture Zones -> Arcs* command will convert the capture zone outline to feature arcs, creating a separate coverage for each well. Coverages can be exported as a shapefile or converted to CAD data. Capture zones are defined using the "Delineate by well" and "Delineate by zone code" options in the MODPATH Display Options dialog.

Pathlines -> Arcs

The *MODPATH | Pathlines -> Arcs* command works similar to the *MODPATH | Capture Zones -> Arcs* but instead of converting the capture zone boundary, all of the pathlines in the capture zone are converted.

Starting Locations -> 3D Scatter Points

Starting locations can be converted to 3D scatter points and vice versa. If the *MODPATH | Starting Locations -> 3D Scatter Points* menu command is selected, a new 3D scatter point set will be created from each particle set that is visible. If the command is executed by right-clicking on a particle set in the Project Explorer, a new 3D scatter point set will be created from only that particle set.

Tab delimited file

Pathlines can be exported by right-clicking on a particle set in the Project Explorer and selecting the *Export* option. This saves the pathline geometry to a tab-delimited text file that can be imported directly into Excel:

	A	B	C	D	E
1	**Particle Index**	**X**	**Y**	**Z**	**Time**
2	1	2759.749707	4673.000293	642.5853882	0
3	1	2778.499707	4675.989063	642.9729004	10.16185188
4	1	2860.999707	4698.607715	646.5800781	59.94756317
5	1	2889.394238	4710.500293	648.5951538	89.09955597
6	1	2951.749707	4762.960742	655.4408569	199.6464691
7	1	2989.360547	4793.000293	659.4976807	277.4127808
8	1	3051.574658	4865.093555	670.4102783	457.3159485
9	1	3070.031934	4883.750293	673.9478149	514.0853882
10	1	3152.77583	4983.575488	695.2171631	833.0230103
11	1	3161.382275	4995.122363	698.9933472	871.7883301
12	1	3183.180371	5020.475391	706.703064	971.0857544
13	2	2758.831982	4678.794238	642.5853882	0
14	2	2778.499707	4688.078906	643.0006714	10.88638687
15	2	2842.216748	4710.500293	645.999939	45.19052124
16	2	2860.999707	4734.410449	649.6905518	72.89888
17	2	2931.588574	4793.000293	655.0541382	189.3552246
18	2	2951.749707	4821.638965	659.7767944	242.1230316
19	2	3009.110303	4883.750293	667.6638794	399.1149292
20	2	3051.574658	4943.287891	678.0623169	554.4108887
21	2	3087.816357	4983.575488	686.4725342	689.6051636
22	2	3146.907178	5056.291797	706.2245483	954.8782959

Spreadsheet report

You can bring up a spreadsheet report showing the pathline XYZ data in GMS. See MODPATH Particle Sets, Spreadsheet Report.

6.5. MT3DMS

MT3DMS

MT3DMS	
Model Info	
Model type	3D Transport Model
Developer	-
Documentation	MT3DMS Manual [1] MT3DMS v5.3 Supplemental User's Guide [2]
Tutorials	MT3DMS Tutorials

MT3DMS is a modular three-dimensional transport model for the simulation of advection, dispersion, and chemical reactions of dissolved constituents in groundwater systems (Zheng, 1990). MT3DMS uses a modular structure similar to the structure utilized by MODFLOW. MT3DMS is used in conjunction with MODFLOW in a two step flow and transport simulation. Heads and cell-by-cell flux terms are computed by MODFLOW during the flow simulation and are written to a specially formatted file. This file is then read by MT3DMS and utilized as the flow field for the transport portion of the simulation.

MT3DMS is a newer version of the MT3D model distributed with earlier versions of GMS. MT3DMS differs from MT3D in that it allows for multi-species transport, supports additional solvers, and allows for cell-by-cell input of all model parameters.

A complete description of MT3DMS is beyond the scope of this reference manual. It is assumed that the reader has a basic knowledge of MT3DMS and has read the MT3DMS documentation (Zheng, 1990). Only the details of the GMS graphical interface to MT3DMS are described herein.

GMS supports MT3DMS as a pre- and post-processor. The input data for MT3DMS is generated by GMS and saved to a set of files. These files are then read by MT3DMS when MT3DMS is executed. MT3DMS can be launched from the GMS menu. The output from MT3DMS is then imported to GMS for post-processing.

A special version of MT3DMS is distributed with GMS. This version of MT3DMS has been modified to output GMS data set files.

The RT3D, SEAM3D and PHT3D models are special versions of MT3DMS that have been customized to simulate reactive transport problems. The interfaces to MT3DMS, RT3D, SEAM3D and PHT3D are all contained in the *MT3D* menu.

MT3DMS Links

- MT3D Packages
- Building an MT3DMS Simulation
- Saving an MT3DMS Simulation
- Importing an MT3D Simulation

References

[1] http://gmsdocs.aquaveo.com/mt3dms.pdf
[2] http://gmsdocs.aquaveo.com/mt3dms_v5_supplemental.pdf

MT3D Packages

MT3DMS is divided into a series of components called "packages." Each package performs a specific task. Some of the packages are always required for a simulation, and some are optional. The input for each package is contained in a separate text file. The MT3DMS packages supported in the GMS interface are listed in the following table:

Package Name	Abrev Name	Decription	Always Req'd?
Basic Transport Package	BTN	Handles basic tasks that are required by the entire transport model. Among these tasks are definition of the problem, specification of the boundary and initial conditions, determination of the step size, preparation of mass balance information, and printout of the simulation results.	YES
Advection	ADV	Solves the concentration change due to advection with one of the three mixed Eulerian-Langrangian schemes included in the package: MOC, MMOC, or HMOC	NO
Dispersion	DSP	Solves the concentration change due to dispersion with the explicit finite difference method.	NO
Sink & Source Mixing	SSM	Solves the concentration change due to fluid sink/source mixing with the explicit finite difference method. Sink/source terms may include wells, drains, rivers, recharge, and evapotranspiration. The constant-head boundary and general-head-dependent boundary are also handled as sink/source terms in the transport model.	NO
Chemical Reactions	RCT	Solves the concentration change due to chemical reactions. Currently, the chemical reactions include linear or nonlinear sorption isotherms and first-order irreversible rate reactions (radioactive decay or biodegradation).	NO
Generalized Conjugate Gradient Solver	GCG	This package can be used to implicitly solve the dispersion, source/sink, and reaction terms of the transport equation.	NO
Transport Observation	TOB	Outputs concentration at observation points and mass fluxes at groups of source/sink boundary conditions.	NO
PHT3D-PHREEQC Interface	PHC	Used by PHT3D to define options for species related to PHREEQC geochemical reactions.	NO

MT3D Source/Sink Mixing Package

The *Source/Sink Mixing Package* dialog is used to assign concentrations to point sources/sinks (wells, river/streams, specified head, general head) and areal sources/sinks (recharge, evapotranspiration). In most cases, the concentration is assigned to the incoming water when the sources/sinks are acting in the source mode, i.e., contributing water to the system. In sink mode, the concentration of the outgoing water is equal to the concentration of groundwater in the aquifer and the concentrations assigned to the sources/sinks have no effect. The only exception is evapotranspiration. Even though evapotranspiration always acts in sink mode, a concentration can be specified. Generally, the assigned concentration is zero which only allows pure water to leave the aquifer.

Source/Sink Mixing Package dialog

Maximum number of Sources/Sinks in Flow Model

MT3DMS needs to know the maximum number of sources/sinks in the flow model. This number is simply the total number of cells in the MODFLOW model that have a source or sink defined at them. This number is computed automatically.

Point sources/sinks spreadsheet

Values for the concentration as well as the type of source or sink and the location of each of the point source/sink cells are displayed and edited in the spreadsheet. Since the concentration data are time dependent, one value is defined for the concentration at each stress period. Concentration values are displayed in the spreadsheet for the current stress period only. The current stress period is displayed above the spreadsheet and can be changed. If the concentration values for a previous stress period are the same as those used for the current stress period, the **Use previous** option may be used. When the **Use previous** option is selected, the fields in the spreadsheet cannot be edited.

A column is provided for each of the species concentrations. Changing the source/sink type for one stress period at a cell will automatically change the type for the remainder of the stress periods at the same cell.

Point sources/sinks can be added and removed from the list by selecting the **Add** and **Delete** buttons. However, when adding new point source/sink, it is usually more convenient to select the 3D grid cells and use the ***MT3D\|Point Sources/Sinks*** menu command.

Initializing Point Source Sinks from MODFLOW

An option is included in the *Source/Sink Mixing Package* dialog to initialize the point sources/sinks to be used in the MT3DMS simulation directly from data that have already been defined in a MODFLOW simulation. If a MODFLOW simulation is currently in memory, the buttons in the *Initialize point sources/sinks from MODFLOW* dialog can be used. Each button is labeled with a different source/sink type, corresponding to the MODFLOW point sources or sinks. Selecting one of the buttons automatically adds to the list of point sources/sinks. For example, if the Well button is selected, GMS creates a new well source/sink for each cell where a well is defined in the MODFLOW simulation currently in memory. The initial concentration of each of the new well sources/sinks is zero. In order to utilize the source/sink initialization buttons, the following conditions must be met:

1. The MODFLOW package corresponding to the source/sink type to be initialized must have been selected in the MODFLOW Packages dialog.
2. Point sources or sinks of the type to be initialized must have been defined in MODFLOW.

Specified Concentration Boundary Conditions

When entering the concentrations for a specified concentration cell, it is sometimes necessary to specify the concentrations of some of the species but allow the concentrations for the remaining species to vary. This can be accomplished by specifying a negative concentration for the species that are to vary.

Areal Sources/Sinks

Recharge and evapotranspiration are known as areally distributed sources and sinks. This is because a value for concentration must be entered for each species for every vertical column of cells in the finite difference grid. This essentially means that a two dimensional array of concentration values must be defined for each species for both recharge and evapotranspiration, depending upon which options are utilized. The buttons labeled Recharge and Evapotranspiration bring up the Areal Source/Sink Array dialog.

Reset

The Reset button deletes all of the data currently defined in the *Source/Sink Mixing package* and restores the package parameters to the default values.

Conceptual Model Input

In some cases, the simplest way to define both point and areal sources/sinks is with an MT3DMS conceptual model in the Map module. Concentrations can be assigned directly to points, arcs, and polygons; points representing point sources/sinks and polygons representing recharge and evapotranspiration zones. These concentrations are copied directly to the appropriate cells/arrays in the *Source/Sink Mixing package* when the conceptual model is converted to the numerical model with the ***Feature Objects | Map -> MT3DMS*** command.

Basic Transport Package Dialog

The first step in setting up an MT3DMS simulation is to define the data for the Basic Transport package. The information defined in the Basic Transport package includes the computational time intervals (stress periods), an array defining which cells are inactive and which cells have constant concentration, an array defining aquifer porosity, and array of starting concentration values. The input data for the Basic Transport package must be entered before editing any of the other packages in the MT3DMS simulation. The *Basic Transport Package* dialog contains the following options:

Headings

A brief description of the model can be entered in the two lines provided at the top of the Basic Transport Package dialog. This information is printed to the ASCII listing file output by MT3DMS.

Model Selection

The Model section in the middle of the *Basic Transport Package* dialog is used to select which transport model is to be used. The menu commands and packages and options that are available depend on which model is selected. All MT3D-based transport models are contained in the *MT3D* menu. See the RT3D, SEAM3D and PHT3D pages for more information on those transport models.

Stress Periods

As is the case with MODFLOW, the computational time intervals for an MT3DMS simulation are called "stress periods". Concentrations at boundary conditions or source/sink terms can only change at the beginning of each stress period. Stress periods are subdivided into time steps and time steps are subdivided into transport steps. The **Stress Periods** button on the left of the *Basic Transport Package* dialog is used to bring up the *Stress Period* dialog. If a transient MODFLOW simulation is used, the stress periods and time steps used for MT3DMS are initialized to coincide exactly with those defined for MODFLOW. If a steady state MODFLOW simulation is used, any set of stress periods may be utilized for MT3DMS.

The *Stress Periods* dialog is identical to the *MODFLOW Stress Periods* dialog except for two fields. In addition to the stress period definition required by MODFLOW, MT3DMS also requires a transport step size and a maximum number of transport steps allowed for one time step. These values are defined for each stress period. If a value of zero is entered for the transport step size (the default), MT3DMS will automatically calculate an appropriate transport step size. Furthermore, if the value for the transport step size entered by the user is larger than the value computed by MT3DMS, the MT3DMS value will be used. The transport step size times the Max transport steps must be longer than the total simulation length.

Output Control

Options for printing and saving the results from an MT3DMS simulation are also included in the Basic Transport package. The MT3DMS output control options are modified by selecting the Output Control button on the left side of the *Basic Transport Package* dialog. This brings up the *MT3D Output Control* dialog.

One of the output options is an unformatted (binary) concentration file which is used for post-processing by GMS. This option is selected using the toggle at the bottom of the dialog. With the version of MT3DMS included with GMS, the concentration file is saved directly from MT3DMS as a GMS binary scalar data set file.

Another option is to Print or save at specified times. When this option is selected, the Times button is undimmed. Selecting the **Times** button brings up a spreadsheet that enables users to specify the times that the output will be printed or saved. The spreadsheet can be quickly initialized by selecting the **Initialize Values** button at the bottom of the dialog. This button brings up the *Initialize Time Steps* dialog. This dialog is composed of edit fields that will automatically populate the times that MT3D data will be saved or printed.

Packages

The **Packages** button on the left of the Basic Transport Package dialog brings up the *Packages* dialog. This dialog is used to specify which of the packages are to be used in the simulation. The check box to the left of the package name is selected to signify that a package will be utilized as part of the simulation. Some of the packages are used by RT3D or SEAM3D and are dimmed for MT3DMS simulations.

Define Species

MT3DMS is a multi-species transport model. It can track the migration and concentration of several species at once. The **Define Species** button is used to define the number of species in the simulation and the name and type of each species. The button brings up the *Define Species* dialog.

The species are listed in the box on the left side of the dialog. Species are added and deleted using the **New** and **Delete** buttons. The name of a selected species can be edited. The **Mobile** toggle is used for RT3D and is dimmed for MT3DMS simulations.

The species names are not used by the MT3DMS code. They are only used in GMS to simplify the model input. In MT3DMS, all species are identified by an integer ID. The species names are saved to the MT3DMS super file (*.mts). When building a new simulation, it is sometimes useful to use the same set of names used in a previous

simulation. This can be accomplished by selecting the Import button and selecting the MT3DMS super file used by the previous simulation. This automatically loads in the species names to the list.

Units

All MT3DMS input parameters must be entered using a consistent set of units. The Units button brings up a dialog which can be used to specify a standard unit for length, time, mass, force, and concentration. The selected units are used by GMS to post the appropriate unit labels next to each of the input fields in the MT3DMS interface. These labels serve as a reminder to the user of the correct units. The units must be consistent with the units used in the MODFLOW simulation.

CINACT

The CINACT value is written to the MT3DMS solution file wherever an inactive concentration cell exists (ICBUND=0). This value should be selected so that it will not likely be a valid concentration computed from MT3DMS. The default value of -999 is generally sufficient.

Use Materials for Porosity and Long. Dispersivity

If this option is selected, the porosities and longitudinal dispersivities specified in the Materials dialog will be used in the MT3DMS model.

HTOP Equals Top of Layer 1

In some cases it is useful to explicitly define the HTOP array separately from the top elevation array for layer 1 defined in the Global Options package. If the toggle entitled HTOP equals top of layer 1 is turned off, an HTOP array can be explicitly defined. This option should be used if the top of layer 1 is substantially higher than the computed water table. In such cases, using the top of layer 1 could lead to significant error (see MT3DMS Reference Manual).

Another option would be to use the default approach and lower the value of the layer 1 top elevation array. For unconfined layers, this array is not used by MODFLOW anyway. The only reason to keep the HTOP array separate from the top elevation array is for visualization. When a MODFLOW solution is displayed in side view, GMS plots the computed water table on top of the cross section display. Using the top elevation of layer 1 as the ground surface makes it possible to see exactly where the water table lies in relation to the ground surface and illustrates where cells are flooded.

HTOP and Thickness Arrays

Part of the input to the Basic Transport package is a set of arrays defining the layer geometry. These arrays include an HTOP array that defines the top elevations for the top layer and a thickness array for each layer. Since MT3DMS can only be used in combination with the True Layer approach in MODFLOW, there is no need to input these arrays in the MT3DMS interface. By default, the HTOP array is assumed to be equal to the top elevation array for layer 1 defined in the Global Options package of MODFLOW. Furthermore, the thickness arrays are automatically generated by GMS using the top and bottom elevation arrays when the Basic Transport package file is written.

ICBUND

The ICBUND button in the Basic Transport Package dialog is used to enter the values of the ICBUND array.

Porosity

The array defining the porosity of each cell in the model can be defined and edited by selecting the **Porosity** button in the Basic Transport Package. This array can also be initialized using a conceptual model in the Map module. It can also be edited on a cell-by-cell basis using the ***Cell Properties*** command.

ICBUND Array

The **ICBUND** button in the Basic Transport Package dialog is used to enter the values of the ICBUND array. The MODFLOW Array Editor is used to edit the ICBUND array.

Values

The ICBUND array contains a value for each cell in the grid defining the type of the cell as either constant concentration, inactive, or variable concentration.

Constant Concentration

A negative ICBUND value indicates that the cell has a constant concentration. The value of the constant concentration is defined in the starting concentration array described below.

Inactive

An ICBUND value of zero indicates that the cell is inactive.

Variable Concentration

A positive ICBUND value indicates that the cell has a variable concentration (i.e., the concentration will be computed as part of the simulation).

Initializing the ICBUND Array

The IBOUND array used by MODFLOW is similar (but not identical) to the ICBUND array. When a new MT3DMS simulation is initialized, the MODFLOW IBOUND array is used to initialize the values of the ICBUND array. Each cell which is designated as inactive in the IBOUND array is designated as inactive in the ICBUND array. Each cell which is designated as active (variable head) in the IBOUND array is designated as active (variable concentration) in the ICBUND array. Each cell that is designated as constant head in the IBOUND array is NOT designated as constant concentration in the ICBUND array. Rather, constant head cells are designated as active (variable concentration) in the ICBUND array.

Cell Attributes Command

In addition to directly editing the ICBUND array, another method for editing the ICBUND array is to select a set of cells and use the ***Cell Properties*** command in the *MT3D* menu.

Activate Cells in Coverage Command

If the ***Activate Cells in Coverage*** command in the Map module is used to change the active/inactive status of the cells, both the IBOUND and ICBUND arrays are updated accordingly.

Flow vs. Transport Inactive Zones

Since the computational domain of the transport simulation does not necessarily have to match the domain of the flow simulation, the active/inactive zones of the ICBUND array may differ from the active/inactive zones of the IBOUND array. In some cases, the active zone for a transport simulation is only a subset of the active zone of the flow simulation. Care should be taken, however, to ensure that any cells defined as inactive in the flow simulation are not defined as active in the transport simulation.

IBOUND vs. ICBUND Display

If a cell is designated as inactive, it is hidden when the grid is displayed, unless the Inactive cells option is selected in the *3D Grid Display Options* dialog, in which case the inactive cells are displayed, but in a different color. If neither MODFLOW nor MT3DMS is in memory, the default 3D grid active/inactive flags are used to control the display of the grid. If MODFLOW is in memory, the active/inactive status of the cells as defined by the IBOUND array is used to display the grid. If both MODFLOW and MT3DMS are in memory, the ICBUND array takes precedence and is used to display the grid.

MT3D Source/Sink Mixing Package

The *Source/Sink Mixing Package* dialog is used to assign concentrations to point sources/sinks (wells, river/streams, specified head, general head) and areal sources/sinks (recharge, evapotranspiration). In most cases, the concentration is assigned to the incoming water when the sources/sinks are acting in the source mode, i.e., contributing water to the system. In sink mode, the concentration of the outgoing water is equal to the concentration of groundwater in the aquifer and the concentrations assigned to the sources/sinks have no effect. The only exception is evapotranspiration. Even though evapotranspiration always acts in sink mode, a concentration can be specified. Generally, the assigned concentration is zero which only allows pure water to leave the aquifer.

Source/Sink Mixing Package dialog

Maximum number of Sources/Sinks in Flow Model

MT3DMS needs to know the maximum number of sources/sinks in the flow model. This number is simply the total number of cells in the MODFLOW model that have a source or sink defined at them. This number is computed automatically.

Point sources/sinks spreadsheet

Values for the concentration as well as the type of source or sink and the location of each of the point source/sink cells are displayed and edited in the spreadsheet. Since the concentration data are time dependent, one value is defined for the concentration at each stress period. Concentration values are displayed in the spreadsheet for the current stress period only. The current stress period is displayed above the spreadsheet and can be changed. If the

concentration values for a previous stress period are the same as those used for the current stress period, the **Use previous** option may be used. When the **Use previous** option is selected, the fields in the spreadsheet cannot be edited.

A column is provided for each of the species concentrations. Changing the source/sink type for one stress period at a cell will automatically change the type for the remainder of the stress periods at the same cell.

Point sources/sinks can be added and removed from the list by selecting the **Add** and **Delete** buttons. However, when adding new point source/sink, it is usually more convenient to select the 3D grid cells and use the ***MT3D\Point Sources/Sinks*** menu command.

Initializing Point Source Sinks from MODFLOW

An option is included in the *Source/Sink Mixing Package* dialog to initialize the point sources/sinks to be used in the MT3DMS simulation directly from data that have already been defined in a MODFLOW simulation. If a MODFLOW simulation is currently in memory, the buttons in the *Initialize point sources/sinks from MODFLOW* dialog can be used. Each button is labeled with a different source/sink type, corresponding to the MODFLOW point sources or sinks. Selecting one of the buttons automatically adds to the list of point sources/sinks. For example, if the Well button is selected, GMS creates a new well source/sink for each cell where a well is defined in the MODFLOW simulation currently in memory. The initial concentration of each of the new well sources/sinks is zero. In order to utilize the source/sink initialization buttons, the following conditions must be met:

1. The MODFLOW package corresponding to the source/sink type to be initialized must have been selected in the MODFLOW Packages dialog.
2. Point sources or sinks of the type to be initialized must have been defined in MODFLOW.

Specified Concentration Boundary Conditions

When entering the concentrations for a specified concentration cell, it is sometimes necessary to specify the concentrations of some of the species but allow the concentrations for the remaining species to vary. This can be accomplished by specifying a negative concentration for the species that are to vary.

Areal Sources/Sinks

Recharge and evapotranspiration are known as areally distributed sources and sinks. This is because a value for concentration must be entered for each species for every vertical column of cells in the finite difference grid. This essentially means that a two dimensional array of concentration values must be defined for each species for both recharge and evapotranspiration, depending upon which options are utilized. The buttons labeled Recharge and Evapotranspiration bring up the Areal Source/Sink Array dialog.

Reset

The Reset button deletes all of the data currently defined in the *Source/Sink Mixing package* and restores the package parameters to the default values.

Conceptual Model Input

In some cases, the simplest way to define both point and areal sources/sinks is with an MT3DMS conceptual model in the Map module. Concentrations can be assigned directly to points, arcs, and polygons; points representing point sources/sinks and polygons representing recharge and evapotranspiration zones. These concentrations are copied directly to the appropriate cells/arrays in the *Source/Sink Mixing package* when the conceptual model is converted to the numerical model with the ***Feature Objects | Map -> MT3DMS*** command.

MT3D point Sources/Sinks BC

The *Source/Sink Mixing Package* dialog can be used to edit both point and areal sources/sinks using a spreadsheet. In many cases, it is more convenient to view and edit source sink concentrations on a cell-by-cell basis. Two commands are provided to facilitate this type of editing: the ***Point Sources/Sinks*** command and the ***Areal Sources/Sinks*** command.

Point Sources/Sinks

The ***Point Sources/Sinks*** command is used to assign and edit the concentrations at point sources and sinks. Before selecting the ***Point Sources/Sinks*** command, a set of cells should be selected using the cell selection tools. Once the command is selected, the *MT3DMS Point Source/Sink* dialog appears.

A point source/sink concentration for each species is assigned to the selected cells by selecting the toggle corresponding to the type of source/sink. The concentration at the cells can be specified as either constant or transient. If a constant value is specified, this number will be used for all stress periods. If the Variable option is used, clicking on the window brings up the XY Series Editor. The XY Series Editor is used to assign a concentration value for each stress period.

Multiple Sources/Sinks per Cell

In many cases, multiple point sources/sinks, each having a different type, can be assigned to an individual cell. In this case, the toggle for each type should be selected and the concentration specified. It is also possible for multiple sources/sinks of the same type to be assigned to a single cell. However, MT3DMS only allows one concentration to be assigned to each type for a single cell. In such cases, the specified concentration for that type applies to all sources/sinks of that type within the cell.

Fixing Concentrations for Selected Species

When entering the concentrations for a specified concentration cell, it is sometimes necessary to specify the concentrations of some of the species but allow the concentrations for the remaining species to vary. This can be accomplished by specifying a negative concentration for the species that are to vary.

Areal Sources/Sinks

The ***Areal Sources/Sinks*** command is used to edit the concentrations assigned to recharge and evapotranspiration fluxes. Before selecting the ***Areal Sources/Sinks*** command, a set of cells should be selected using the cell selection tools. The concentration fluxes due to recharge and evapotranspiration are applied to vertical columns rather than to individual cells. Therefore, to edit the value for a vertical column, any cell in the column can be selected. Once the ***Areal Sources/Sinks*** command is selected, the *MT3DMS Areal Sources/Sinks* dialog appears.

The areal source/sink concentration values at the selected cells can be edited by highlighting a species and either selecting a constant value for all stress periods, or by defining a set of values.

Building an MT3DMS Simulation

Setting up an MT3DMS simulation involves taking a pre-defined MODFLOW simulation and defining some additional properties such as porosity, assigning concentrations to sources and sinks, and choosing some general simulation options. MODFLOW automatically generates a head and flow file (*.hff) containing information on computed heads and fluxes. This file is used as part of the input to the transport model when MT3DMS is launched. MT3DMS uses the top and bottom grid elevation specified in the *Global Options/Basic Package* Dialog. A MODFLOW model must exist in the GMS project prior to creating an MT3DMS simulation.

Two basic approaches are provided in GMS for defining these data: using the 3D Grid module or using the Map module.

Using the 3D Grid Module

Although it is not always the most efficient approach, an MT3DMS simulation can be completely defined using only the tools in the 3D Grid module. With this approach, the material properties and concentrations at sources/sinks are assigned directly to the cells.

Using the Map Module (MT3DMS Conceptual Model)

For sites with complicated boundary conditions and sources/sinks, the preferred method for setting up an MT3DMS simulation is to use the feature object tools in the Map module to define an MT3DMS conceptual model of a site being studied.

In order to use the conceptual model approach to build a MT3DMS model, the same conceptual model must be used for both the MODFLOW flow model and the MT3DMS transport model. This ensures that there is a proper linkage between the sources/sinks in the conceptual model and the sources/sinks in the grid model.

The conceptual model is a high-level description of the site describing sources/sinks, the boundary of the domain to be modeled, recharge and evapotranspiration zones, and material zones within each of the layers. In addition to the parameters required by MODFLOW, many of the parameters required by MT3DMS such as concentrations at sources/sinks and layer data, including porosity and dispersion coefficients, can be assigned directly to the feature objects.

The following steps are used in setting up a conceptual model and converting the conceptual model to a numerical model:

1. Construct a MODFLOW conceptual model, create a grid, and convert the conceptual model data to the MODFLOW data defined at the grid cells.
2. Run the MODFLOW simulation to save the MT3DMS head and flow file.
3. Return to the Map module. Edit the properties of the Conceptual model to include transport. Assign concentrations to the sources/sinks in the conceptual model where necessary. Also define polygonal zones describing layer data including porosity, longitudinal dispersivity, sorption constants, rate constants, and bulk density.
4. Select the ***Feature Objects | Map -> MT3DMS*** command to automatically assign the MT3DMS data to the appropriate cells in the grid.

Saving an MT3DMS Simulation

Once an MT3DMS simulation has been created and checked for potential problems with the Model Checker, the final step is to save the simulation to disk and run MT3DMS. MT3DMS simulations are saved using the ***Save*** and ***Save As*** commands in the ***File*** menu.

Selecting the Save command saves the GMS project including the model simulation. By default the model simulation will be saved to the same location as the GMS project. However, in the Save dialog the path for the model simulation can be specified to be different from the project file if the "Match model names with project name" toggle is unchecked. This is done with the Browse buttons next to each model.

An MT3DMS simulation is saved to a set of input files. The MT3DMS super file is a special type of file which is used to organize the set of files used in a simulation. The names of all of the input and output files associated with a simulation are saved in the super file. When MT3DMS is launched, the name of the super file is automatically passed to the MT3DMS executable.

When an MT3DMS simulation is saved, the names of the other MT3DMS input files are automatically patterned after the name of the super file. For example, if the super file is named sampmod.mts, the other files are named sampmod.btn, sampmod.ssm, etc.

This content also applies to RT3D, SEAM3D and PHT3D.

Importing an MT3D Simulation

It is often necessary to import an MT3DMS simulation that was not generated by GMS. Since GMS uses the standard MT3DMS file format, this is not a problem in most cases. However, there are a few steps and precautions that should be taken.

File Formats

GMS uses the standard MT3DMS file formats described in the MT3DMS documentation (Zheng, 1998). The files you are importing must match these formats exactly. If the files were generated for a version of MT3DMS that uses a different set of file formats, the files will need to be edited before they can be successfully imported to GMS.

CON File

The source code of MT3DMS was changed to create a *.con file instead of the default *.ucn file. GMS can read either the *.con or *.ucn file. The *.con file is written in the GMS binary data set format.

Importing Super Files

GMS uses an MT3DMS super file to organize the files used by an MT3DMS simulation. This file is not a standard MT3DMS file. One approach to importing an externally defined simulation is to create an MT3DMS super file containing the names of the files used in the simulation using a text editor.

MT3DMS Display Options

The properties of all MT3DMS data that GMS displays on the screen can be controlled through the MT3D tab of the *Display Options* dialog. This dialog is opened by right clicking on the 3D Grid Data entry in the Project Explorer and selecting the Display Options command. It can also be accessed from the from the *Display* menu, the *MT3D* menu, or the **Display Options** macro. The following table describes the display options available for the 3D Grid module.

Display Option	Description
Point Sources/Sinks, NAPL Point Sources/Sinks, Constant concentration	The three items at the top of the dialog represent display options for the constant concentration cells and the point sources/sinks. If the check box just to the left of one of these is selected, a symbol is displayed at the center of each cell where that object has been defined. The symbol for each object is displayed in a window to the left of the check box. The symbol can be changed by clicking on the button. This brings up the *Symbol Editor* dialog. The Symbol Editor contains a list of available symbols and can be used to edit the size and color of the symbol.
Display symbol legend	If the Symbol legend option is selected, a legend showing each of the symbols associated with sources/sinks and constant concentration cells is displayed in the lower right corner of the GMS window.
Check All	By clicking this button, all of the display options are turned on.
Check None	By clicking this button, all of the display options are turned off.

Calculate Mass

The *MT3D Calculate Mass* dialog allows the calculation of the mass of a contaminant in the entire model domain or in the selected cells.

Definition

Explains the formula that is used to calculate the mass. Basically, the volume of water in each cell is calculated and then multiplied by the concentration computed by MT3D. A conversion factor must be supplied so that the calculated mass will be computed in the correct units.

Concentration Range

Allows the specification a range over which the mass will be calculated. This is most useful to ignore negative concentrations that result in MT3D simulations.

Conversion Factor

The factor determines which units the resulting mass will be displayed in.

6.6. PEST

PEST

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

PEST is a general purpose parameter estimation utility developed by John Doherty of Watermark Computing. The PEST interface in GMS can be used to perform automated parameter estimation for MODFLOW. The steps involved in setting up a PEST run are described in the automated parameter estimation topic.

The PEST options are edited by selecting the ***Parameter Estimation*** command in the *MODFLOW* menu and then clicking on the PEST tab.

PEST version support

The following table shows the version of PEST supported by different versions of GMS. To see the latest released version of PEST visit http://www.pesthomepage.org/Downloads.php.

GMS	PEST
GMS 6.5	PEST 10.0
GMS 7.0	PEST 10.0
GMS 7.1	PEST 12.0
GMS 8.0, 8.1, 8.2	PEST 12.1
GMS 8.3, 9.0	PEST 12.2
GMS 9.1	PEST 12.3

6.7. PHT3D

PHT3D

PHT3D	
Model Info	
Model type	geochemical, multi-species reactive transport
Developer	Henning Prommer, Vicent Post
Documentation	PHT3D website [1] PHT3D Manual [2]
Tutorials	GMS:Tutorials

PHT3D is a multicomponent transport model for three-dimensional reactive transport in saturated porous media developed and maintained by Henning Prommer and Vicent Post. PHT3D is a combination of MT3DMS and PHREEQC-2. The PHREEQC component allows for a variety of low temperature aqueous geochemical reactions.

Basic Transport Package

To use PHT3D, the user selects *PHT3D* as the Model in the *Basic Transport Package* dialog.

Define Species

The options that are specific to PHT3D are entered when defining the species for the simulation. When the **Define Species** button is selected the *PHT3D Options* dialog comes up. In addition to defining the species for the simulation, the user can enter options specific to PHT3D (such as temperature, activation criterion etc.) See the PHT3D documentation for more information on specific variables in this dialog.

PHT3D requires as part of its input a PHREEQC database file. The species that can be included in the simulation depend on the contents of the PHREEQC database file. The browser button near the top of the *PHT3D Options* dialog allows the user to select a PHREEQC database file. GMS will parse the contents of this file and make available the species listed in the file. The user also has the option to select a PHT3D-PHREEQC interface file. When an interface package file is chosen GMS assumes that the PHREEQC database file is in the same directory with the name pht3d_datab.dat. In addition to parsing the PHREEQC database, the interface file is also read and any options that are specific to the species are imported.

The PHREEQC database file can contain many species. When a user is only using a small subset of the available species in the database it is useful to only show the species that are active (in use). The **Only show active species** check box hides all of the species that are not active.

All other inputs to PHT3D are the same as MT3DMS. For more information on how to use PHT3D the user is referred to the GMS tutorials.

References

[1] http://pht3d.org

[2] http://gmsdocs.aquaveo.com/PHT3D_manual_v210.pdf

6.8. RT3D

RT3D

RT3D	
Model Info	
Model type	multi-species reactive transport
Developer	Battelle Pacific Northwest National Laboratory
Documentation	RT3D Manual v1 [1] RT3D Manual v2.5 [2]
Tutorials	RT3D Tutorials

RT3D is a multi-species reactive transport model developed by the Battelle Pacific Northwest National Laboratory. RT3D is a modified version of MT3DMS that utilizes alternate Chemical Reaction packages. Numerous pre-defined reactions are available and an option is provided for creating user-defined reactions. RT3D is well-suited for simulating natural attenuation and bioremediation.

Since RT3D is a modified version of MT3DMS, most of the input to RT3D is identical to the input required for MT3DMS. Thus, the RT3D interface is contained within the *MT3D* menu in the 3D Grid module. In the *Basic Transport Package* dialog, an option is provided for selecting the current model as either MT3DMS, RT3D, or SEAM3D. A number of options in the interface then change based on which model is selected.

Since much of the RT3D interface is identical to the MT3DMS Interface, only the portions of the interface which are unique to RT3D are described in this help file.

Basic Transport Package

The first step in defining an RT3D simulation is to define the data required by the Basic Transport (BTN) package. The options in the *Basic Transport Package* dialog unique to RT3D are as follows:

Packages

The Packages button brings up the *Packages* dialog. If the RT3D model is the current model and the Chemical Reaction package is selected, one of the RT3D reactions must be selected from the pull-down list. The first nine reactions are pre-defined reaction types. If one of these reactions is selected, the names of the species and the names of the reaction parameters are automatically determined by GMS. The last reaction is a user-defined reaction. If this option is selected, a list of species and list of reaction parameters must be specified by the user.

Define Species

For most of the reaction package options, once the reaction package is selected, the list of species used by the package is automatically initialized by GMS. However, if the user-defined reaction package is selected, the **Define Species** button is undimmed in the *Basic Transport Package* dialog and a list of species must be manually defined before any concentrations are assigned to sources/sinks.

Chemical Reactions Package

The Chemical Reactions Package dialog utilized by RT3D is different from the MT3DMS Chemical Reactions dialog. The items in the dialog unique to RT3D are as follows:

Solver

If the selected reaction package is a kinetic reaction, several solvers are available for the solution of the chemical reaction equations. The desired solver should be selected from the Solver pull-down list. When one of these solvers is used, an absolute and relative tolerance must be specified for each of the mobile species using the atol and rtol parameters.

Reaction Parameters

With each reaction package, a set of reaction parameters must be defined. The method used to edit the reaction parameters depends on whether the reaction is a pre-defined reaction or a user-defined reaction.

Pre-Defined Reactions

For the pre-defined reactions, the number of reaction parameters and the names of the parameters are fixed. If the Spatially vary all toggle is off, a single value is entered for each parameter using the edit field below the parameter list. If the **Spatially vary all** toggle is on, an array of values is entered for each parameter using the **Edit** button. In this case, the cell-by-cell parameter values can also be edited using the ***Cell Attributes*** command.

User-Defined Reactions

For user-defined reactions, the list of reaction parameters must be defined using the **Define Parameters** button. This button brings up the Define Parameters dialog. This dialog functions similarly to the *Define Species* dialog. The New and Delete buttons are used to add and remove items from the list. The **Import** button is used to read a previously defined list of reaction parameters from an RT3D super file (*.rts).

With the pre-defined reactions, the reaction parameters are either all spatially variable or all constant. However, with the user-defined reaction option, selected parameters may be designated as spatially variable while others are designated as constant. The variable/constant status of a parameter is selected using the Spatially variable toggle in the *Define Parameters* dialog.

References

[1] http://gmsdocs.aquaveo.com/rt3dman.pdf
[2] http://gmsdocs.aquaveo.com/rt3dv2.5man.pdf

Basic Transport Package Dialog

The first step in setting up an MT3DMS simulation is to define the data for the Basic Transport package. The information defined in the Basic Transport package includes the computational time intervals (stress periods), an array defining which cells are inactive and which cells have constant concentration, an array defining aquifer porosity, and array of starting concentration values. The input data for the Basic Transport package must be entered before editing any of the other packages in the MT3DMS simulation. The *Basic Transport Package* dialog contains the following options:

Headings

A brief description of the model can be entered in the two lines provided at the top of the Basic Transport Package dialog. This information is printed to the ASCII listing file output by MT3DMS.

Model Selection

The Model section in the middle of the *Basic Transport Package* dialog is used to select which transport model is to be used. The menu commands and packages and options that are available depend on which model is selected. All MT3D-based transport models are contained in the *MT3D* menu. See the RT3D, SEAM3D and PHT3D pages for more information on those transport models.

Stress Periods

As is the case with MODFLOW, the computational time intervals for an MT3DMS simulation are called "stress periods". Concentrations at boundary conditions or source/sink terms can only change at the beginning of each stress period. Stress periods are subdivided into time steps and time steps are subdivided into transport steps. The **Stress Periods** button on the left of the *Basic Transport Package* dialog is used to bring up the *Stress Period* dialog. If a transient MODFLOW simulation is used, the stress periods and time steps used for MT3DMS are initialized to coincide exactly with those defined for MODFLOW. If a steady state MODFLOW simulation is used, any set of stress periods may be utilized for MT3DMS.

The *Stress Periods* dialog is identical to the *MODFLOW Stress Periods* dialog except for two fields. In addition to the stress period definition required by MODFLOW, MT3DMS also requires a transport step size and a maximum number of transport steps allowed for one time step. These values are defined for each stress period. If a value of zero is entered for the transport step size (the default), MT3DMS will automatically calculate an appropriate transport step size. Furthermore, if the value for the transport step size entered by the user is larger than the value computed by MT3DMS, the MT3DMS value will be used. The transport step size times the Max transport steps must be longer than the total simulation length.

Output Control

Options for printing and saving the results from an MT3DMS simulation are also included in the Basic Transport package. The MT3DMS output control options are modified by selecting the Output Control button on the left side of the *Basic Transport Package* dialog. This brings up the *MT3D Output Control* dialog.

One of the output options is an unformatted (binary) concentration file which is used for post-processing by GMS. This option is selected using the toggle at the bottom of the dialog. With the version of MT3DMS included with GMS, the concentration file is saved directly from MT3DMS as a GMS binary scalar data set file.

Another option is to Print or save at specified times. When this option is selected, the Times button is undimmed. Selecting the **Times** button brings up a spreadsheet that enables users to specify the times that the output will be printed or saved. The spreadsheet can be quickly initialized by selecting the **Initialize Values** button at the bottom of

the dialog. This button brings up the *Initialize Time Steps* dialog. This dialog is composed of edit fields that will automatically populate the times that MT3D data will be saved or printed.

Packages

The **Packages** button on the left of the Basic Transport Package dialog brings up the *Packages* dialog. This dialog is used to specify which of the packages are to be used in the simulation. The check box to the left of the package name is selected to signify that a package will be utilized as part of the simulation. Some of the packages are used by RT3D or SEAM3D and are dimmed for MT3DMS simulations.

Define Species

MT3DMS is a multi-species transport model. It can track the migration and concentration of several species at once. The **Define Species** button is used to define the number of species in the simulation and the name and type of each species. The button brings up the *Define Species* dialog.

The species are listed in the box on the left side of the dialog. Species are added and deleted using the **New** and **Delete** buttons. The name of a selected species can be edited. The **Mobile** toggle is used for RT3D and is dimmed for MT3DMS simulations.

The species names are not used by the MT3DMS code. They are only used in GMS to simplify the model input. In MT3DMS, all species are identified by an integer ID. The species names are saved to the MT3DMS super file (*.mts). When building a new simulation, it is sometimes useful to use the same set of names used in a previous simulation. This can be accomplished by selecting the Import button and selecting the MT3DMS super file used by the previous simulation. This automatically loads in the species names to the list.

Units

All MT3DMS input parameters must be entered using a consistent set of units. The Units button brings up a dialog which can be used to specify a standard unit for length, time, mass, force, and concentration. The selected units are used by GMS to post the appropriate unit labels next to each of the input fields in the MT3DMS interface. These labels serve as a reminder to the user of the correct units. The units must be consistent with the units used in the MODFLOW simulation.

CINACT

The CINACT value is written to the MT3DMS solution file wherever an inactive concentration cell exists (ICBUND=0). This value should be selected so that it will not likely be a valid concentration computed from MT3DMS. The default value of -999 is generally sufficient.

Use Materials for Porosity and Long. Dispersivity

If this option is selected, the porosities and longitudinal dispersivities specified in the Materials dialog will be used in the MT3DMS model.

HTOP Equals Top of Layer 1

In some cases it is useful to explicitly define the HTOP array separately from the top elevation array for layer 1 defined in the Global Options package. If the toggle entitled HTOP equals top of layer 1 is turned off, an HTOP array can be explicitly defined. This option should be used if the top of layer 1 is substantially higher than the computed water table. In such cases, using the top of layer 1 could lead to significant error (see MT3DMS Reference Manual).

Another option would be to use the default approach and lower the value of the layer 1 top elevation array. For unconfined layers, this array is not used by MODFLOW anyway. The only reason to keep the HTOP array separate from the top elevation array is for visualization. When a MODFLOW solution is displayed in side view, GMS plots the computed water table on top of the cross section display. Using the top elevation of layer 1 as the ground surface makes it possible to see exactly where the water table lies in relation to the ground surface and illustrates where cells are flooded.

HTOP and Thickness Arrays

Part of the input to the Basic Transport package is a set of arrays defining the layer geometry. These arrays include an HTOP array that defines the top elevations for the top layer and a thickness array for each layer. Since MT3DMS can only be used in combination with the True Layer approach in MODFLOW, there is no need to input these arrays in the MT3DMS interface. By default, the HTOP array is assumed to be equal to the top elevation array for layer 1 defined in the Global Options package of MODFLOW. Furthermore, the thickness arrays are automatically generated by GMS using the top and bottom elevation arrays when the Basic Transport package file is written.

ICBUND

The ICBUND button in the Basic Transport Package dialog is used to enter the values of the ICBUND array.

Porosity

The array defining the porosity of each cell in the model can be defined and edited by selecting the **Porosity** button in the Basic Transport Package. This array can also be initialized using a conceptual model in the Map module. It can also be edited on a cell-by-cell basis using the ***Cell Properties*** command.

6.9. SEAM3D

SEAM3D

SEAM3D	
Model Info	
Model type	reactive transport
Developer	Mark Widdowson at Virginia Tech University
Documentation	SEAM3D Manual [1]
Tutorials	MODAEM Tutorials

SEAM3D is a reactive transport model used to simulate complex biodegradation problems involving multiple substrates and multiple electron acceptors. It is based on the MT3DMS code. In addition to the regular MT3DMS modules, SEAM3D includes a Biodegradation package and NAPL Dissolution package. SEAM3D was developed by Mark Widdowson at Virginia Tech University.

Since SEAM3D is a modified version of MT3DMS, most of the input to SEAM3D is identical to the input required for MT3DMS. Thus, the SEAM3D interface is contained within the MT3D menu in the 3D Grid module. In the *Basic Transport Package* Dialog, an option is provided for selecting the current model as either MT3DMS, RT3D, or SEAM3D. A number of options in the interface then change based on which model is selected.

References

[1] http://gmsdocs.aquaveo.com/SEAM3Dv3.pdf

6.10. SEAWAT

SEAWAT

SEAWAT	
Model Info	
Model type	3D Flow and Transport Model
Developer	USGS
Documentation	SEAWAT Manual [1]
Tutorials	SEAWAT Tutorials

SEAWAT is a three dimensional variable density groundwater flow and transport model developed by the USGS based on MODFLOW and MT3DMS. SEAWAT v4 is based on MODFLOW 2000 and MT3DMS 5.2. SEAWAT includes two additional packages: Variable-Density Flow (VDF) and Viscosity (VSC). A complete description of SEAWAT is beyond the scope of this reference manual. It is assumed that the reader has a basic knowledge of SEAWAT and has read the SEAWAT documentation.

GMS supports SEAWAT as a pre- and post-processor. The interface to SEAWAT relies on the interface to MODFLOW and MT3DMS. The input data for SEAWAT is generated by GMS and saved to a set of files including a MODFLOW model, an MT3D model if transport is used, and a SEAWAT model pointing to the MODFLOW and MT3D model's package files. These files are then read by SEAWAT and executed. SEAWAT uses the MODFLOW and MT3D interfaces for boundary condition display and for post-processing.

SEAWAT Links

- Building a SEAWAT Model
- SEAWAT Global Options Dialog
- SEAWAT MODFLOW AUX Variables
- SEAWAT Packages
- Importing a SEAWAT Simulation
- MODFLOW with HDF5

References

[1] http://gmsdocs.aquaveo.com/seawatv4.pdf

Building a SEAWAT Model

SEAWAT	
Model Info	
Model type	3D Flow and Transport Model
Developer	USGS
Documentation	SEAWAT Manual [1]
Tutorials	SEAWAT Tutorials

Grid Approach

Although it is not always the most efficient approach, a SEAWAT simulation can be completely defined using only the tools in the 3D Grid module. To build a SEAWAT model with this approach you must first create a MODFLOW model using the grid approach. If using transport, an MT3D model is then created, and then a SEAWAT model is created. For each of the 3 models the properties and concentrations at sources/sinks assigned directly to the cells. Any SEAWAT MODFLOW AUX Variables are assigned using the MODFLOW package dialogs. SEAWAT specific packages can be set using the VDF or VSC package dialogs.

Conceptual Model Approach

To create a SEAWAT using a conceptual model:

1. Construct a MODFLOW conceptual model, create a grid, and convert the conceptual model data to the MODFLOW data defined at the grid cells.
2. Run the MODFLOW simulation to make sure it is working properly.
3. If the simulation includes transport, return to the Map module. Edit the properties of the Conceptual model to include transport. Assign concentrations to the sources/sinks in the conceptual model where necessary. Also define polygonal zones describing layer data including porosity, longitudinal dispersivity, sorption constants, rate constants, and bulk density.
4. Select the *Feature Objects | Map -> MT3DMS* command to automatically assign the MT3DMS data to the appropriate cells in the grid.
5. Enable SEAWAT in the MODEL interfaces dialog and create a SEAWAT simulation.
6. Edit the MODFLOW AUX variables, VDF, and VSC packages if necessary.
7. Use the SEAWAT model checker to check the simulation for any errors or warnings.

SEAWAT Global Options Dialog

SEAWAT	
Model Info	
Model type	3D Flow and Transport Model
Developer	USGS
Documentation	SEAWAT Manual [1]
Tutorials	SEAWAT Tutorials

The *Global Options* Dialog is used to setup a new SEAWAT model or to edit the current options for an existing model. To set up a new SEAWAT model select Edit|Model Interfaces... menu and turn on the SEAWAT model interface. Next select SEAWAT|New menu item to create the SEAWAT model.

In the Global Options dialog, toggles are provided to modify the use of transport in the simulation, to turn on and off the Variable-Density Flow (VDF) and Viscosity (VSC) packages, and to enable or disable the SEAWAT MODFLOW AUX Variables variables used by SEAWAT.

SEAWAT MODFLOW AUX Variables

SEAWAT	
Model Info	
Model type	3D Flow and Transport Model
Developer	USGS
Documentation	SEAWAT Manual [1]
Tutorials	SEAWAT Tutorials

SEAWAT allows using MODFLOW AUX variables for some packages to set values used for MODFLOW packages the variables include:

- WELDENS - Density of well fluid (WEL package).
- DRNBELEV - Drain bottom elevation (DRN package).
- RBDTHK - River bed thickness (RIV package).
- RIVDEN - Density of river fluid (RIV package).
- GHBELEV - General head elevation (GHB package).
- GHBDENS - Density of general head fluid (GHB package).
- CHDDENSOPT and CHDDEN - Constant head fluid density (CHD package).

These AUX variables can be enabled by selecting them in the SEAWAT Global Options Dialog and can be edited in the associated MODFLOW package dialog or in the MODFLOW Sources/Sinks dialog.

SEAWAT Packages

SEAWAT	
Model Info	
Model type	3D Flow and Transport Model
Developer	USGS
Documentation	SEAWAT Manual [1]
Tutorials	SEAWAT Tutorials

SEAWAT specific packages include the Variable-Density Flow (VDF) and Viscosity (VSC) packages. The package dialogs for both packages are available under the SEAWAT menu. For the package menu items to be available, the packages must first be enabled from the SEAWAT Global Options Dialog. For a detailed description of the VDF and VSC input options, please refer to the SEAWAT manual available in the side bar.

VDF Package

The VDF package can be used in a SEAWAT simulation to model the affects of concentration on fluid density. The input values available in the VDF package include:

- IWTABLE - Active variable-density water table corrections.
- MFNADVFD - Internodal density calculation method.
- DENSEMIN - Minimum fluid density.
- DENSEMAX - Maximum fluid density.
- FIRSTDT - Length of first transport time step.
- NSWTCPL - Flow/transport coupling.
- DNSCRIT - Convergence criteria.
- MT3DRHOFLG - type of fluid density calculation.
  - When equal to zero, the fluid density for each grid cell can be entered in the table at the bottom of the dialog. For INDENSE enter -1 to use values from previous stress period, 0 to use DENSREF, and 1 to enter values for DENSE column.
  - When equal to -1, the fluid density will be calculated using one or more species entered in the table at the bottom of the dialog using species ID, DRHODC, and CRHOREF. Rows can be entered into the table with the tool bar below the table.
  - When greater than or equal to 1 density is calculated using the corresponding species number.
- DENSREF - Reference fluid density.
- DRHODC - Density/conc. slope.
- DRHODPRHD - Density/pressure slope.
- PRHDREF - Reference pressure head.

VSC Package

The VSC package can be used in a SEAWAT simulation to model the affects of concentration on fluid viscosity. Using the Viscosity Package requires using the VDF package and the LPF package in MODFLOW. The input values available in the VSC package include:

- VISCMIN - Minimum viscosity.
- VISCMAX - Maximum viscosity.
- MT3DMUFLG - Type of viscosity calculation.
  - When equal to zero, the fluid viscosity for each grid cell can be entered in the table at the bottom of the dialog. For INVISC enter -1 to use values from previous stress period, 0 to use VISCREF, and 1 to enter values for VISC column.
  - When equal to -1, the fluid density will be calculated using one or more species entered in the table at the bottom of the dialog using species ID, DMUDC, and CMUREF. Rows can be entered into the table with the tools bar below the table.
  - When greater than or equal to 1 viscosity is calculated using the corresponding species number.
- VISCREF - Reference viscosity.
- DMUDC - Viscosity/concentration slope.
- CMUREF - Reference concentration.
- MTMUTEMPSEC - Temp. species ID.
- AMUCOEFF - coefficients for equation 18-20 in SEAWAT 4 documentation.

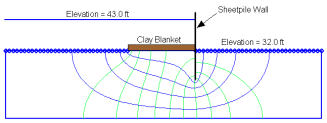
SEAWAT Importing a SEAWAT Simulation

SEAWAT	
Model Info	
Model type	3D Flow and Transport Model
Developer	USGS
Documentation	SEAWAT Manual [1]
Tutorials	SEAWAT Tutorials

An existing SEAWAT v4 model can be imported into GMS by opening the name file which must have a "*.nam" extension. Upon opening the name file GMS uses the name file to determine if the simulation is MODFLOW-2000, MODFLOW-2005, or a SEAWAT simulation based on the packages included. Once the import is complete, if the model includes transport packages or the VDF or VSC packages then a SEAWAT model is automatically created inside GMS. If the model only includes MODFLOW packages, then the SEAWAT model needs to be created by enabling the SEAWAT model interface by selecting SEAWAT in the Model Interfaces dialog available under the Edit menu, and then selecting ***New Simulation*** under the *SEAWAT* menu.

6.11. SEEP2D

SEEP2D

SEEP2D	
Sample Confined Seepage Problem  *SEEP2D Screenshot*	
Model Info	
Model type	2D seepage, confined or unconfined, steady state
Developer	Fred Tracy, ERDC [1]
Documentation	SEEP2D Primer [2]
Tutorials	SEEP2D Tutorials

SEEP2D is a two-dimensional steady state finite element groundwater model developed by Fred Tracy of the Engineer Research and Development Center [1]. Both saturated and unsaturated flow is simulated. SEEP2D is designed to be used on profile models (XZ models) such as cross-sections of earth dams or levees.

A variety of options are provided in GMS for displaying SEEP2D results. Contours of total head (equipotential lines) and flow vectors can be plotted. An option is also available for computing flow potential values at the nodes. These values can be used to plot flow lines. Together with the equipotential lines (lines of constant total head), the flow lines can be used to plot a flow net. The phreatic surface can also be displayed.

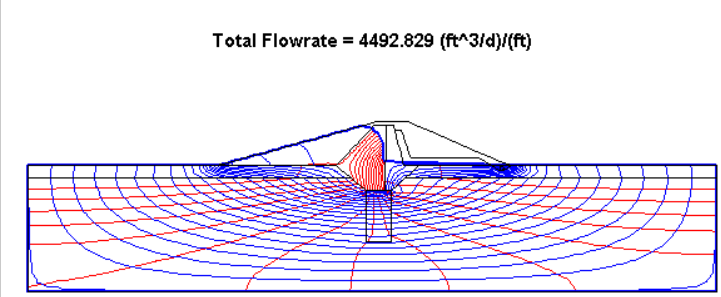
A more complete description of the SEEP2D model, including a discussion of boundary conditions and guidelines for model conceptualization is contained in the SEEP2D Primer. The SEEP2D Primer should be reviewed before consulting this help file. The user is also encouraged to complete the SEEP2D tutorials.

Portions of a SEEP2D model can be imported from GeoStudio files.

Building a SEEP2D Simulation

For a typical application, the following steps are used to perform a SEEP2D simulation using GMS:

1. **Generate a 2D mesh** - The first step in setting up a SEEP2D simulation is to construct a 2D finite element mesh. This can be accomplished using the ***Feature Objects | Map -> 2D Mesh*** command in the Map module. SEEP2D only supports



Sample flow net from SEEP2D

meshes composed entirely of linear elements (three node triangles and four node quadrilaterals). The mesh should not contain quadratic elements (six node triangles and eight node quadrilaterals).

2. **Select the analysis options** - The analysis options are used to set up the type of problem being modeled and to defined constants that are used by SEEP2D when calculating the solution. The items in the dialog are as follows:
   - *Title* - A descriptive title can be entered for the simulation. This title is used in the header of the SEEP2D input and output files. It can also be displayed at the top of the Graphics Window in GMS by turning on the **Title** option in the *SEEP2D Display* Options dialog.
   - *Datum* - By default, the datum of the model is at zero, but it can be specified to any convenient value, such as the value corresponding to the base or lowest y coordinate of the model.
   - *Unit Weight of Water* - The unit weight of water must be entered. SEEP2D uses this value to compute pore pressures. The weight and length units defined in this value should be consistent with the units used elsewhere in the model.
   - *Units* - The **Units** button brings up the *Units* dialog. This dialog is used to enter the units for length, time, concentration, etc. for the simulation. GMS uses the selected unit options to display the appropriate units next to each input edit field in the other SEEP2D dialogs.
   - *Problem Type* - The problem type must be specified either as plane flow or axisymmetric flow. The axisymmetric option should be selected for models corresponding to flow to a single well as described in the SEEP2D Primer. All other models should use the plane flow option.
   - *Flow Lines* - If the Compute flow lines option is turned on, once the head solution is computed, SEEP2D will reverse the boundary conditions and compute flow potential values at the nodes. These values can be contoured by GMS using the Flow lines option is the *SEEP2D Display Options* dialog.
   - *Model Type* - The model type should be selected as either Confined or Unconfined. For confined models, the entire model domain is assumed to be saturated. No exit face boundary conditions should be applied and the unsaturated zone material properties are not required. For unconfined models, two options are available for dealing with the unsaturated zone: (1) deforming mesh and (2) saturated/unsaturated flow modeling. For both types of problems, exit face boundary conditions should be applied along the boundary of the mesh where the free surface is expected to exit. With the deforming mesh option, SEEP2D iterates to find the location of the phreatic surface and the mesh is deformed or truncated so that the upper boundary of the mesh matches the phreatic surface. The solution files from this type of simulation include a geometry file containing the deformed mesh. With the saturated/unsaturated option, the mesh is not modified and the flow in both the saturated and unsaturated zone is modeled. The hydraulic conductivity in the unsaturated zone is modified (reduced) using either the linear frontal method or the Van Genuchten method. The equations used by both methods are described in more detail in the SEEP2D Primer.
3. **Assign the material properties** - Each element in the 2D mesh is assigned a material ID. The material properties are assigned to each element using these IDs and a list of material properties. The material properties specific to SEEP2D are entered using the ***Material Properties*** command in the *SEEP2D* menu. This command brings up the *SEEP2D Material Properties* dialog. The items in the dialog are as follows:
   - *List of Materials* - The currently defined materials are listed at the top of the dialog. The values for a material are entered by selecting the material and editing the values in the lower part of the dialog.
   - *Soil Coefficients* - The hydraulic conductivity in the two major principal directions and the angle from the x-axis to the major principal axis are entered in the Soil Coefficients section. These hydraulic conductivity values represent the hydraulic conductivity for saturated conditions.
   - *Van Genuchten Parameters* - If the Van Genuchten Saturated/Unsaturated option has been selected in the *SEEP2D Analysis Options* dialog, the Van Genuchten alpha and n-value numbers must be defined.
   - *Linear Front Parameters* - If the Saturated/Unsaturated with Linear Front option has been selected in the *SEEP2D Analysis Options* dialog, the minimum pressure head (ho) and minimum relative conductivity (kro) values must be defined.

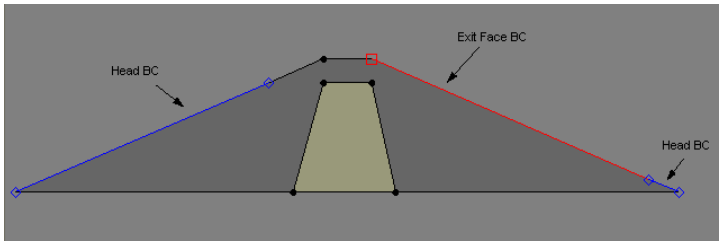
4. **Assign the boundary conditions** (either with a conceptual model or with the 2D mesh tools) - Two general types of boundary conditions can be defined for SEEP2D simulations: nodal boundary conditions and flux boundary conditions. (See SEEP2D Boundary Conditions)
5. **Run the Model Checker.**
6. **Save the simulation**
7. **Run the simulation**
8. **Display the results** - Before reading in the solution file for post-processing in GMS, it is often useful to examine the text output listing file. The output file can also be viewed by simply double-clicking on the *.out file found in the SEEP2D solution folder in the Project Explorer. Also, any text file can be viewed by selecting the ***Edit File*** command in the *File* menu. The File Browser appears and the selected file is opened in a text editor. Once the solution is imported to GMS, a variety of options are available for displaying the solution. The total head and pressure head data sets can be contoured. Velocity vectors corresponding to Darcy velocity can be plotted. If the Compute flow lines option was selected, a complete flow net may be plotted by displaying contours of total head (equipotential lines) and turning on the Flow lines option in the *SEEP2D Display Options* dialog.

SEEP2D Boundary Conditions

Two general types of boundary conditions can be defined for SEEP2D simulations: nodal boundary conditions and flux boundary conditions. The type or value assigned to a previously defined boundary condition can be edited by selecting the node or node string and selecting either the ***Node BC*** or ***Flux BC*** commands. A boundary condition can be deleted by selecting the boundary condition using either the Select Node or Select Node String tools and selecting the ***Delete BC*** command in the *SEEP2D* menu.

Nodal Boundary Conditions

The most common type of SEEP2D boundary conditions is nodal boundary conditions. Nodal boundary conditions are assigned by selecting the nodes and selecting the ***Node BC*** command in the *SEEP2D* menu. When selecting the nodes, either the Select Node tool or the Select Node String tool may be used. The ***Node BC*** command brings up the *Node BC* dialog. Three types of nodal boundary conditions can be assigned: head, exit face, and flow rate.



Sample SEEP2D Problem Showing Boundary Conditions

- **Head BC** - Specified head boundary conditions represent boundaries where the head is known. They typically are found where water is ponding or at the boundary of a region where the water table is known to remain constant. Since the head along such boundaries cannot change, they represent regions of the model where flow enters or exits the system (flow lines are always orthogonal to constant head boundaries).
- **Exit Face BC** - Exit face boundary conditions imply that the head is equal to the elevation (assuming that the datum is 0). They are used when modeling unconfined flow problems and should be placed along the face where the free surface is likely to exit the model. This boundary condition must be used if the option for deforming the mesh to the phreatic surface has been selected in the analysis options. It may also be used with a saturated/unsaturated flow model. In this case, if the head at a node on the boundary becomes greater than the node elevation during the iteration process, the head at the node is fixed at the nodal elevation and the node acts as a specified head boundary. Thus, water is allowed to exit the boundary above the tailwater. If an exit face boundary is not used with a saturated/unsaturated flow model, all of the flow will be forced through the tailwater.

- **Flow Rate BC** - Flow rate boundary conditions are used to specify nodes at which a certain flow rate is known to exist. They are used primarily when modeling wells and the flow specified represents the pumping rate. Negative values represent extraction of fluid from the system whereas positive values represent injection.

Flux Boundary Conditions

Flux boundary conditions are used to specify a known flux rate [L/T] along a sequence of element edges on the perimeter of the mesh. They are often used to simulate infiltration. Flux into the system is positive and flux out of the system is negative.

Flux BC are assigned by selecting a sequence of nodes along the mesh boundary using the Select Node String tool and selecting the ***Flux BC*** command in the *'SEEP2D* menu. This command brings up a simple prompt for the flow rate.

Reading the SEEP2D Solution

Once the simulation is completed, the solution is automatically imported to GMS for post-processing. The solution files are organized in a SEEP2D solution folder in the Project Explorer. If the unconfined/deforming mesh option is selected, the solution will consist of two files: a geometry file that contains the deformed mesh and a solution file that contains the head, velocity, and flow potential data sets. If any of the other analysis options is selected, only the solution file is saved. Regardless of which option is used, the solution can be quickly imported to GMS using the ***Read Solution*** command in the *SEEP2D* menu. The solution data sets are organized into a SEEP2D solution folder in the Data Tree.

SEEP2D Files

See the main page about SEEP2D files at SEEP2D Files.

References

[1] http://www.erdc.usace.army.mil/
[2] http://gmsdocs.aquaveo.com/s2dprimr.pdf

SEEP2D Display Options

The properties of all SEEP2D data that GMS displays on the screen can be controlled through the SEEP2D tab of the *Display Options* dialog. This dialog is opened by right clicking on the 2D Mesh Data entry in the Project Explorer, selecting the ***Display Options*** command, and then selecting the SEEP2D tab. This tab is only visible when there is a SEEP2D simulation. It can also be accessed from the from the ***Display*** menu, the *SEEP2D* menu, or the **Display Options** macro. The following table describes the display options available for the SEEP model.

Display Option	Description
Head BC	The Head BC, Exit face BC, Flux rate BC, and Flux BC items can be used to turn on the display of a symbol for each of the boundary condition types. The color and type of symbol can be edited by clicking on the button to the left of each item.
Exit face BC	
Flow rate BC	
Flux BC	
BC values	If the BC values option is selected, the numerical value of each boundary condition (head, flux rate, etc.) is displayed next to the boundary condition. The font used to display the values can be editing by clicking on the button to the left of the item.
Flow lines	If the Compute flow lines option is selected the SEEP2D Analysis Options dialog prior to saving and running the model, SEEP2D performs the computations in two steps. In the first step, SEEP2D solves for the heads. In the second step, the head solution is used to "reverse" the boundary conditions and a second solution is found. This solution represents "flow potential" values. When the solution is read back into GMS, these flow values can be contoured to generate a plot of flow lines. When superimposed on contours of total head (equipotential lines) a complete flow net can be displayed. (see ***Note***)
Title	If the Title option is selected, the problem title specified in the *Analysis Options* dialog will be displayed at the top of the Graphics Window.
Total flow rate	If the Total flow rate option is selected and if a solution is in memory, the total flow rate through the model will be displayed at the top of the Graphics Window, just below the title.
Phreatic surface	If the Phreatic Surface option is selected and if a solution is in memory, the phreatic surface will be displayed on the mesh. The color and type of line can be edited by clicking on the button to the left of the item.
Check All	By clicking this button, all of the display options are turned on.
Check None	By clicking this button, all of the display options are turned off.

***Note**: When contouring the flow values, GMS must determine a contour interval that will result in the proper number of flow channels. The number of flow channels is computed by solving for numflow in the following equation:*

$$q = k_{\text{equiv}} \frac{\text{numflow}}{\text{numequipotential}} \Delta H$$

The kequiv value is solved for using the k values for the base material specified using the combo box just below the Flow Lines option. The equivalent k is computed as follows:For problems with several material zones where each material is isotropic, the flow net cells in the base material will appear to be square, while the cells in the other material zones will be stretched. The amount of stretching is a function of the relative difference in k values between the material and the base material.

SEEP2D Solution

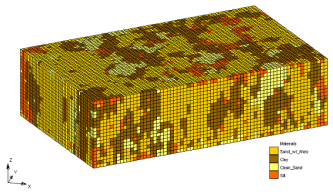
After a SEEP2D simulation is run in GMS, the solution to this simulation can be automatically loaded into GMS. The solution will appear in the Project Explorer under the 2D Mesh Data folder in its own subfolder. The subfolder will appear with a lock image on it. The image means that the information in the solution folder is locked and cannot be edited from within GMS. The solution datasets in the solution can be contoured and viewed in GMS. The potential datasets that may be part of a SEEP2D solution are listed below. A description of each also follows.

Total Head	the pressure head plus the elevation head (same as the head measured by a piezometer).
Pressure Head	the water pressure divided by the unit weight of water.
Pore Pressure	the hydrostatic pressure.
Flowrate	the flow rate of water into (out of) the problem domain.
Velocity	the discharge velocity is calculated by multiplying the gradient by hydraulic conductivity [k*i]. This is a vector data set and is accompanied by a scalar data set called *velocity_Mag* that is the magnitude of the vector data set.
Gradient	the hydraulic gradient is calculated by dividing the difference in Total Head by distance [$(h_1-h_2)/L$]. This is a vector data set and is accompanied by a scalar data set called *gradient_Mag* that is the magnitude of the vector data set.
Flowline	this data set is used to create a flow net when the Total Head data set is the active data set.

- All data set values are calculated at every node in the mesh.

6.12. T-PROGS

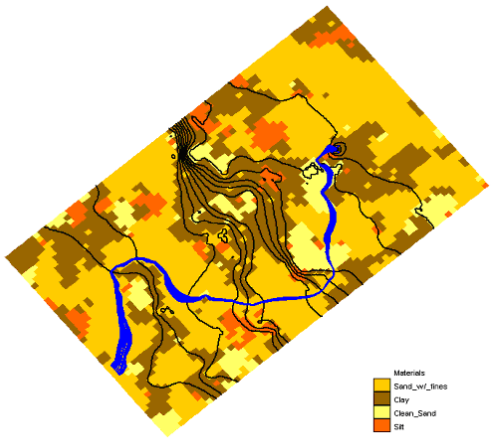
T-PROGS

T-PROGS	
 *T-PROGS Screenshot*	
Model Info	
Model type	transition probability geostatistics on borehole data
Developer	Steven Carle
Documentation	T-Progs Manual [1]
Tutorials	T-PROGS Tutorials

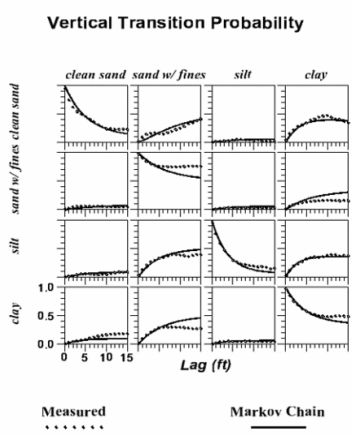
T-PROGS
Links
T-PROGS
Strike Dip Markov Chain
Vertical Markov Chain
TSIM

GMS includes an interface to the T-PROGS software developed by Steven Carle. The T-PROGS software is used to perform transition probability geostatistics on borehole data. The output of the T-PROGS software is a set of N material sets on a 3D grid. Each of the material sets is conditioned to the borehole data and the materials proportions and transitions between the boreholes follows the trends observed in the borehole data. These material sets can be used for stochastic simulations with MODFLOW. A sample material set generated by the TPROGS software is shown below. The T-PROGS software can also be used to generate multiple input datasets for the HUF package.

T-PROGS Interface



MODFLOW solution for a T-PROGS grid. Black lines are head contours and blue lines are pathlines for an extraction well.



Observed transition probabilities (circles) and Markov chain model (solid lines) for a set of borehole data with four materials.

The T-PROGS software utilizes a transition probability-based geostatistical approach to model spatial variability by 3-D Markov Chains, set up indicator co-kriging equations, and formulate the objective function for simulated annealing. The transition probability approach has several advantages over traditional indicator kriging methods. First, the transition probability approach considers asymmetric juxtapositional tendencies, such as fining-upwards sequences. Second, the transition probability approach has a conceptual framework for incorporating geologic interpretations into the development of cross-correlated spatial variability. Furthermore, the transition probability approach does not exclusively rely on empirical curve fitting to develop the indicator (cross-) variogram model. This is advantageous because geologic data are typically only adequate to develop a model of spatial variability in the vertical direction. The transition probability approach provides a conceptual framework to geologic insight into a simple and compact mathematical model, the Markov chain. This is accomplished by linking fundamental observable attributes – mean lengths, material proportions, anisotropy, and juxtapositioning – with Markov chain model parameters.

The first step in using T-PROGS is to import a set of borehole data. The borehole data are then passed to a utility within T-PROGS called GAMEAS that computes a set of transition probability curves as a function of lag distance for each category for a given sampling interval. A sample set of measured transition probability curves are shown by the dashed lines in the following figure.

Each curve represents the transition probability from material j to material k. The transition probability $t_{jk}(h)$ is defined by:

$$t_{jk}(\mathbf{h}) = \Pr\{k \text{ occurs at } \mathbf{x}+\mathbf{h} \mid j \text{ occurs at } \mathbf{x}\}$$

[2]

where x is a spatial location, h is the lag (separation vector), and j,k denote materials. Note that the curves on the diagonal represent auto-transition probabilities, and the curves on the off-diagonal represent cross-transition probabilities.

The next step in the analysis is to develop a Markov Chain model for the vertical direction that fits the observed vertical transition probability data. The Markov Chain curves are shown as solid lines in the preceeding figure. Mathematically, a Markov chain model applied to one-dimensional categorical data in a direction Φ assumes a matrix exponential form:

$$\mathbf{T}(h_\phi) = \exp(\mathbf{R}_\phi h_\phi)$$

[3]

where h denotes a lag in the direction Φ, and RΦ denotes a transition rate matrix

$$\mathbf{R}_\phi = \begin{bmatrix} r_{11,\phi} & \cdots & r_{1k,\phi} \\ \vdots & \ddots & \vdots \\ r_{k1,\phi} & \cdots & r_{kk,\phi} \end{bmatrix}$$

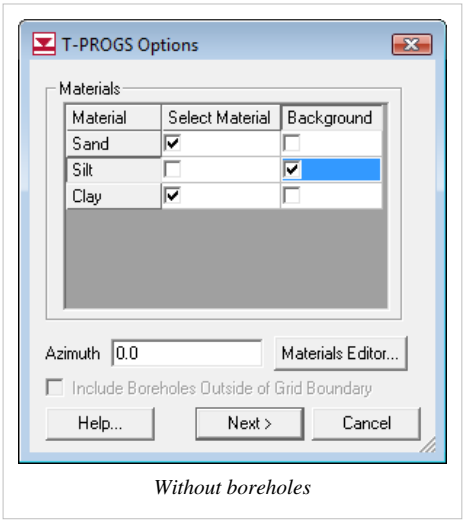
with entries r_{jk},f representing the rate of change from category j to category k (conditional to the presence of j) per unit length in the direction Φ. The transition rates are adjusted to ensure a good fit between the Markov Chain model and the observed transition probability data.

Once the Markov chain is developed for the z direction from the borehole data, a model of spatial variability must be developed for the x and y directions. Borehole data are typically not sufficiently dense in these directions. However, the x and y-direction Markov chains can be developed by assuming that the juxtapositional tendencies and the proportions observed in the vertical direction also hold true in the horizontal directions. The modeler then provides an estimate of the ratio of the mean lengths in the x and y directions relative to the z direction, and the transition rate matrices for the x and y directions can be formulated. The x, y, and z Markov chains are converted into a continuous 3D Markov chain using the MCMOD utility within T-PROGS.

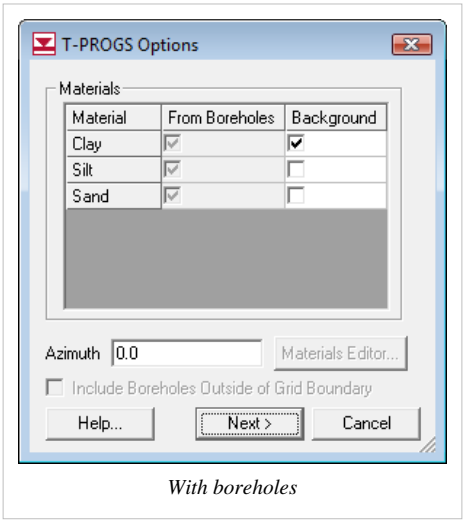
In the final phase of setting up a transition probability analysis using T-PROGS, the modeler creates a grid, specifies the number of model instances (N), and launches the TSIM utility. The TSIM code uses the 3D Markov chain to formulate both indicator cokriging equations and an objective function for simulated annealing. It generates stochastic simulations using a combination of modified versions of the GSLIB codes SISIM and ANNEAL.

T-PROGS OPTIONS

T-PROGS Options dialog with and without borehole data.



Without boreholes



With boreholes

When a user selects the ***New Simulation*** command to initialize a T-PROGS simulation, the *T-PROGS Options* dialog appears. If boreholes do not exist in the model, an unconditioned simulation will be generated. In this case, the user selects the materials to be used and a corresponding background material. The **Materials Editor** button enables users to quickly create, delete, or rename materials and the material list is automatically regenerated. The upper part of the dialog lists the materials in the boreholes. The first column of toggles indicates which materials are to be used in the analysis. By default, all materials associated with the boreholes are selected. These toggles are

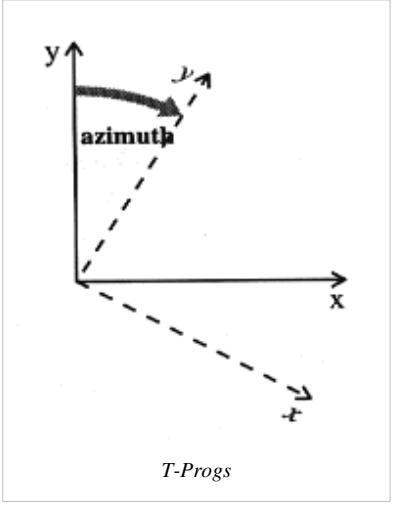
necessary since it is possible that there may be materials defined in the materials list that are not associated with boreholes. The second column in the top section of the dialog lists the background material. By default, the material type that had the predominant occurrence in the boreholes (greatest proportion) is marked as the background material. When defining the transition probability data in the next section, the input parameters do not need to be edited for the background material. The parameters for this material are automatically adjusted to balance the equations.

Background Material

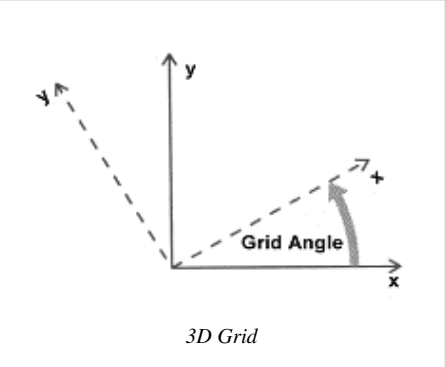
Application of the transition probability approach involves the designation of a background material. The probabilistic constraints of the Markov chains make it unnecessary to quantify data for one category. Not only is it unnecessary, but it is futile to do so because values will be overwritten in order to satisfy constraints. Conceptually, the background material can be described as the material that "fills" in the remaining areas not occupied by other units. For example, in a fluvial depositional system, a floodplain unit would tend to occupy area not filled with higher-energy depositional units and would therefore be a logical choice for the background material.

Azimuth

Azimuths for T-PROGS and 3D Grid.



T-Progs



3D Grid

The user also enters an azimuth in this dialog. The azimuth determines the orientation of the primary directions of the depositional trends in the strike/dip directions. These trends generally are aligned with the primary directions of horizontal flow in the aquifer. Theoretically, the azimuth can be oriented independently from the grid orientation. However, in practice, if the grid and azimuth orientations are offset by more than about 40^{o}, checkerboard patterns appear in the indicator array results. Hence, the azimuth orientation is set equal to the grid orientation by default. However, the grid angle is defined counterclockwise, and the azimuth angle is clockwise. Therefore, if the grid angle is 40^{o}, then the azimuth angle will be -40^{o} by default. If there is anisotropy in the xy plane, the azimuth angle should be set to the principle direction of the anisotropy. If anisotropy is not present, this angle should be coincident with the x-axis (the rows or j-direction) of the grid.

Material Limit

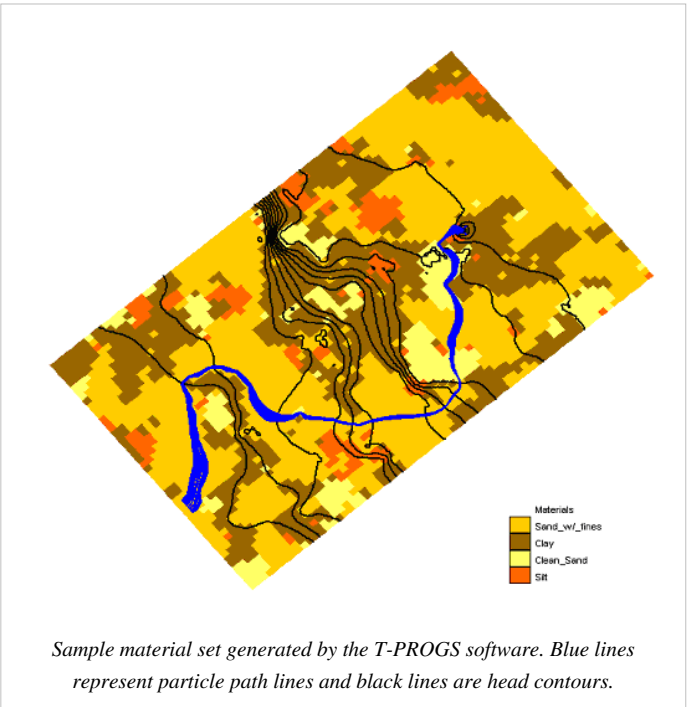
One limitation for both the cases with and without boreholes is that a maximum of five materials can be used in the T-PROGS algorithm. This limitation was imposed to keep the data processing and user-interface reasonably simple. Although five materials present a limitation, borehole data can generally be easily condensed down to five or fewer materials. Furthermore since this is a stochastic approach, which is based on probability, the detail generated with numerous materials is rarely justifiable anyway. In addition, as the number of materials increase, the ratio of process time to detail becomes inefficient.

Generating Material Sets with T-PROGS

The underlying equations solved by the T-PROGS software require an orthogonal grid with constant cell dimensions (X, Y, and Z). The delta X values can be different from the delta Y and delta Z values, and the delta Y values can be different from the delta Z values, but all cells must have the same change in X, Y, and Z dimensions. The MODFLOW model is capable of using the Layer Property Flow (LPF) Package with the Material ID option for assigning aquifer properties. With this option, each cell in the grid is assigned a material id and the aquifer properties (Kh, Kv, etc.) associated with each material are automatically assigned to the layer data arrays for the LPF package when the MODFLOW files are saved. The T-PROGS software generates multiple material sets (arrays of material ids), each of which represents a different realization of the aquifer heterogeneity. When running a MODFLOW simulation in stochastic mode, GMS automatically loads each of the N material sets generated by the T-PROGS software and saves N different sets of MODFLOW input files. The N solutions resulting from these simulations can be read into GMS and used to perform risk analyses such as probabilistic capture zone delineation.

One-Layer MODFLOW Grids

Although MODFLOW is a three-dimensional model, a majority of the MODFLOW models constructed by typical users are 2D models consisting of one model layer. There are several reasons why 2D models are so common. One reason is that many of these models are regional models where the aquifer thickness is very small compared to the lateral extent of the model. As a result, the flow directions are primarily horizontal and little improvement is gained by adding multiple layers to the model. Even with local scale models, the aquifer thickness is often small enough that one-layer models are considered adequate. 2D models are also attractive due to the simplicity of the model increased computational efficiency. One of the problems associated with using multiple layers for MODFLOW models with unconfined aquifers is that as the water table fluctuates, the upper cells may go dry. These cells will not rewet even if the water table subsequently rises, unless the rewetting option has been selected in the flow package (BCF, LPF, or HUF). The rewetting issues can often be avoided with a one-layer model.



Sample material set generated by the T-PROGS software. Blue lines represent particle path lines and black lines are head contours.

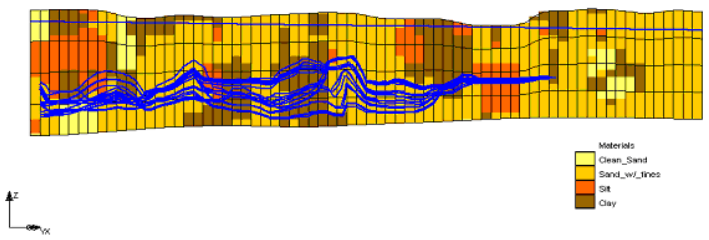
When developing a one-layer model, the modeler must determine how to distribute the hydraulic conductivity values within the layer. One option is to assume a homogenous aquifer; this is typically a gross over-simplification since aquifers are usually highly heterogeneous. Therefore, a common approach is to delineate zones of hydraulic

conductivity by examining the subsurface stratigraphic data. In many cases, these data are in the form of borehole logs. These borehole logs often exhibit substantial heterogeneity and don't always exhibit definitive trends between adjacent boreholes. Furthermore, the boreholes are often clustered with large regions of the model lacking any borehole data. The modeler then faces a difficult task of trying to determine a rational approach to delineating two-dimensional zones of hydraulic conductivity based on complex 3D borehole data.

As part of this research, we developed a technique for developing 2D zones of hydraulic conductivity from borehole logs using transition probability geostatistics. The technique is simple, fast, and preserves proportions and trends exhibited by the borehole data. The algorithm parses through each borehole and computes a predominant material at each borehole. When T-PROGS runs, the predominant material for each borehole is assigned to its corresponding location in the one-layer grid, and during the quenching process, simulations are conditioned to those data points.

Generating HUF Data with T-PROGS

Using transition probability geostatistics with MODFLOW models results in two basic limitations. First, the underlying stochastic algorithms used by the T-PROGS software are formulated such that the MODFLOW grid must have uniform row, column, and layer widths. The row width can be different from the column width, but each row must have the same width. This results in a uniform orthogonal grid. While MODFLOW grids are orthogonal in x and y, the layer thickness is allowed to vary on a cell-by-cell basis. This makes it possible for the layer boundaries to accurately model the ground surface and the tops and bottoms of aquifer units. If a purely orthogonal grid is used, irregular internal and external layer boundaries must be simulated in a stair-step fashion either by varying material properties or by activating/inactivating cells via the IBOUND array. A second limitation is that in order to get a high level of detail in the simulated heterogeneity, the grid cell dimensions are generally kept quite small. This can result in difficulties in the vertical dimension. The large number of layers with small layer thicknesses near the top of the model generally ensures that many of the cells in this region will be at or above the computed water table elevation (for simulations involving unconfined aquifers). As a result, these cells will undergo many of the numerical instabilities and increased computational effort issues associated with cell wetting and drying.



Sample HUF stratigraphy data generated by the T-PROGS software. Blue lines represent particle path lines and water table.

The Hydrogeologic Unit Flow (HUF) package released with MODFLOW 2000 makes it possible to overcome both of these limitations resulting in a powerful mechanism for incorporating transition probability geostatistics in MODFLOW simulations. With the HUF package, the modeler is allowed to input the vertical component of the stratigraphy in a grid-independent fashion. The stratigraphy data are defined using a set of elevation and thickness arrays. The first array defines the top elevation of the model. The remaining arrays define the thicknesses of a series of hydrogeologic units, starting at the top and progressing to the bottom of the model. For each array of thicknesses, many of the entries in the array may be zero. This makes it possible to simulate complex heterogeneity, including pinchouts and embedded lenses that would be difficult to simulate with the LPF and BCF packages.

The T-PROGS interface in GMS includes an option for integrating transition probability geostatistics results with the HUF package. The basic approach used by the option is to overlay a dense background grid on the MODFLOW grid and run T-PROGS on the background grid. A set of HUF arrays is then extracted from the background grid for use with the MODFLOW model. To use this option, user should first create a MODFLOW grid with the desired number of layers and the layer elevations should be interpolated to match the aquifer boundaries. The row and column widths are uniform but the layer thicknesses may vary from cell to cell. Then, when TSIM is launched, the HUF option

should be selected. GMS then generates a background grid that encompasses the MODFLOW grid. The rows and columns of this grid match the MODFLOW grid but the layer thicknesses are uniform and relatively thin, resulting in a much greater number of layers than the MODFLOW grid. The user specifies the number of layers in this background grid. A T-PROGS simulation is then performed to get a set of material sets on the background grid. Each of the material sets in the T-PROGS output is then transferred from the background grid to a set of HUF elevation/thickness arrays. The HUF top elevation array is set equal to the top of the MODFLOW grid. The thickness arrays are then found by searching through the background grid to find the bottom elevations of contiguous groups of indicators. The elevations from these groups are then added to an appropriate elevation array in the HUF input. The resulting set of HUF input arrays are listed in GMS Project Explorer. By clicking on each item in the Project Explorer, the selected set of HUF arrays are loaded into the HUF package and the corresponding stratigraphy is displayed in the GMS window. The multiple HUF input arrays can be used to perform a stochastic simulation.

Notes

[1] http://gmsdocs.aquaveo.com/t-progs.pdf

Vertical Markov Chain

T-PROGS
Links
T-PROGS
Strike Dip Markov Chain
Vertical Markov Chain
TSIM

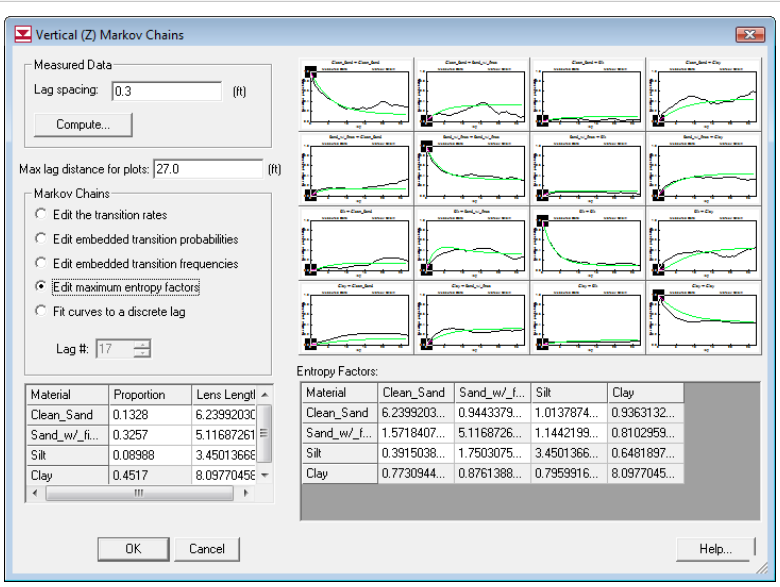
A dialog assists the user in defining the vertical Markov chains. This dialog is composed of three main sections:

- Plot section
- Markov Chains section
- Spreadsheet section

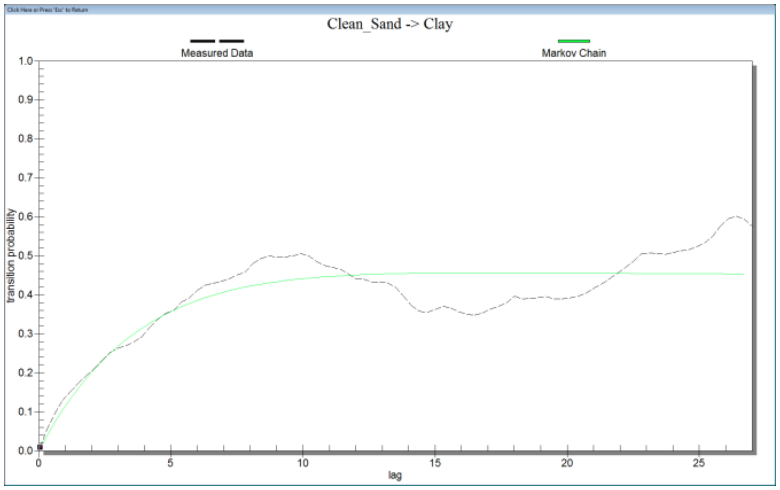
All three sections enable the user to develop a 1-D Markov chain in the vertical direction.

Plot section

The plot section includes the array of curves, the Lag spacing edit field, the Compute button, and the Max lag distance for plots edit field. The number of plots in the array produced correlates to the number of materials used in the simulation. If N materials are used, an N by N array of plots will illustrate the transition probabilities for each material with respect to every other material. Every plot is labeled with a name and units and can be maximized with a command in the menu produced by right-clicking on the curve in question. The curves are automatically regenerated anytime a change is made in the other sections of the dialog.



Vertical Markov chain dialog.



Transition probability plot.

Each of the plots contains two curves depicting the transition probability. The dashed line represents the transition probability measured from the borehole data by the GAMEAS utility. In general, this curve represents the transition probability from material j to material k. The transition probability $t_{jk}(h)$ is defined by:

$$t_{jk}(\mathbf{h}) = \Pr\{k \text{ occurs at } \mathbf{x}+\mathbf{h} \mid j \text{ occurs at } \mathbf{x}\} \quad [1]$$

where x is a spatial location, h is the lag (separation vector), and j,k denote materials. The lag is defined by the Lag spacing item in the upper left corner of the *Vertical (Z) Markov Chains* dialog. The curve shown with the solid line is called a “Markov Chain”. The Markov Chains are used to formulate the equations used by T-PROGS to generate the multiple material sets during the simulation stage. The objective of this stage of the analysis is to fit the Markov Chain curves as accurately as possible to the measured transition probability curves. This process is similar to fitting a model variogram to an experimental variogram in a kriging exercise. The transition rates are adjusted to ensure a good fit between the Markov Chain model and the observed transition probability data.

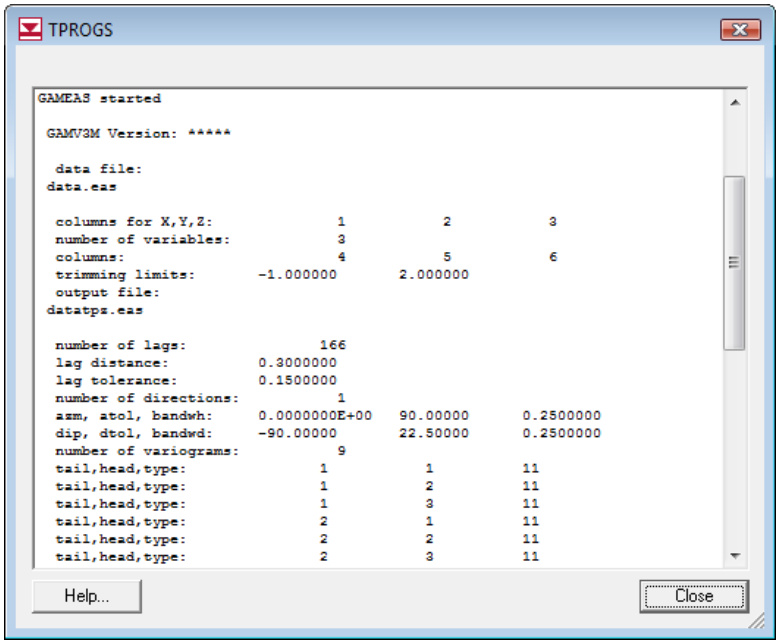
It should be noted that the self-transitional curves on the diagonal start at a probability of 1.0 and decrease with distance and the off-diagonal curves start at zero probability and increase with distance. In both cases, the curves eventually flatten out at some distance. The probability corresponding to the flat part of the curve represents the mean proportion of the material. All curves on a particular column should flatten out to the same proportion. The proportions are displayed in the lower left corner of the dialog. The point where a tangent line from the early part of the curves on the diagonal intersects the horizontal (lag distance) axis on each curve represents the mean lens length for the material. The mean lens lengths are shown just to the right of the mean proportions in the lower left part of the dialog. The slope at the beginning of each of the Markov Chains represents the transition rate. Together, the proportions, lens lengths, and transition rates define the Markov Chains.

Several methods are provided for fitting the Markov Chains to the measured transition probability curves. These methods are listed in the section of the dialog titled *Markov Chains*. By default, GMS automatically makes an attempt to fit the curves using the **Edit maximum entropy factors** option. In many cases, this fit is sufficiently accurate and we can proceed to the next step. However, it is often useful to explore the other options for fitting the curves.

If boreholes exist in the model, the **Compute...** button becomes undimmed. When the user clicks the **Compute...** button, the parameter files required for running GAMEAS are generated and GAMEAS is executed.

GAMEAS

GAMEAS is an algorithm that processes borehole data and determines geologic characteristics such as material proportions and transition probability curves in a given direction. A separate window displays the details of the GAMEAS run, which allows the user to monitor the output from the GAMEAS simulation.



GAMEAS executable inside GMS interface.

When GAMEAS completes a successful run, the results, including the material proportions and transition probability curves from the measured data, are read into the corresponding data fields in the *Vertical Markov Chain* dialog. Furthermore, the transition rates which correspond to the slope of the transition probability curves when the lag equals 0 are interpolated from the measured data curves. In addition to running GAMEAS, the code parses through the borehole data and calculates the embedded transition probabilities and frequencies. These values are stored in arrays that correspond to options #2 and #3 in the Markov Chains section of the dialog.

Lag

The Lag spacing determines how dense the curves are, and the Max lag distance for plots determines the range of the curves. The curves always range from 0 to the Max lag distance for plots horizontally, and they range from 0 to 1 vertically to honor probability constraints.

Markov chains section

There are five alternate methods of generating Markov chains in this section (area in the *Vertical Markov Chain* dialog). These methods are in a radio-button configuration to allow the user to conveniently change from one method to another. Each of these five methods will be described.

- Edit the transition rates
- Edit embedded transition probabilities
- Edit embedded transition frequencies
- Edit maximum entropy factors
- Fit curves to a discrete lag
- Edit the transition rates

Edit the transition rates

With this option one can directly edit the array of transition rates that are listed in the Transition Rates section. This option is useful after selecting the Compute button and running GAMEAS because slopes can be inferred from the measured data curves. GAMEAS outputs transition probability curves. Transition rates used in this option correspond to the slope of the transition probability curve at a lag = 0. When reading the output from GAMEAS, the transition probability rates are interpolated as

$$r_{jk,\phi} = 0.57 * r1_{jk,\phi} + 0.29 * r2_{jk,\phi} + 0.14 * r3_{jk,\phi}$$

where r1, r2, and r3 are the slopes defined by a straight line from the origin out to lag1, lag2, and lag3 respectively. As the lag approaches zero, more weight should be given to the corresponding slope. Hence, a weight of 0.57, 0.29, and 0.14 were assigned to r1, r2, and r3 respectively. Once the slopes are computed for each entry in the matrix, the mean lengths for each category are computed by

$$\bar{L}_{j,\phi} = \frac{-1}{r_{jj,\phi}}$$

Regardless of which Markov Chain option is selected, the background row and column, Sand_w/_fines, is dimmed because the values in this row and column are automatically computed from the remaining entries by probability constraints of the background material. In addition, with this option selected, the Lens Length column is also dimmed because the lens lengths are automatically computed and updated from the diagonal terms in the Transition Rates spreadsheet. The diagonal terms of the Transition Rates spreadsheet must be negative to obey probability rules. With this data, this method produces an accurate fit between the measured (green) and the Markov chain (blue) curves at small lag spaces.

Edit the embedded transition probabilities

This is a more intuitive method of generating Markov chains and is conducive to sites with and without data. It is conducive to sites with data because the embedded transition probabilities can be determined from the borehole data. When a simulation is initialized, if borehole data exist default embedded transition probabilities are computed from the borehole data. If borehole data do not exist, the embedded transition probabilities can be estimated with some basic geologic knowledge including the average mean lengths of each material for each direction and depositional trends.

With this option selected, the diagonal terms are dimmed because these values are derived from the values entered in the Lens Length column. Adjusting the proportions, lens lengths, or the off-diagonal terms in the Transition Rates spreadsheet alters the curves.

With this option, the spreadsheet in the lower right of the dialog represents the probability that the row material will transition to the column material moving vertically upwards in the +Z direction.

Edit the embedded transition frequencies

This option is similar to option #2, except embedded transition frequencies populate the off-diagonal terms of the Transition Rates spreadsheet. Embedded transition frequencies are also computed from borehole data when a simulation is initialized if borehole data exist. Once again, this is an intuitive framework with geologic knowledge including average mean lengths and depositional trends.

Edit maximum entropy factors

While this option can be used with borehole data, it is ideally suited for cases without borehole data. With this option, the user edits the proportions for all but the background material and the mean lens lengths for all materials. The lens lengths are used to populate the diagonal terms of the Transition Rates spreadsheet, and the maximum entropy factors fill the off-diagonal terms of the spreadsheet. The maximum entropy factors represent the ratio of the transition rate to the maximum entropy transition rate. A maximum entropy factor of 1.0 represents maximum disorder in depositional tendencies. A rate greater than 1.0 indicates that the two categories tend to occur next to each other. A factor less than unity would infer the opposite. This is an intuitive method of generating Markov chains and is conducive to all types of sites. This method enables logical incorporation of anisotropy into the model with the maximum entropy factors.

Fit curves to a discrete lag

This option is only undimmed if a transition probability curve from measured data exists in memory. When this option is selected, the **Lag #** edit field is undimmed and the user enters the discrete lag the curves will be fit to.

This option produces Markov chains that are computed from the measured transition rates produced by GAMEAS. GAMEAS computes a set of transition probabilities at each lag specified by the user. This option computes an array of transition rates from the slope of the curves generated by GAMEAS (green). The transition rates correlate to the shape of the curve from the origin to the lag # specified by the user in the Lag # edit field. Therefore, the cells of all the spreadsheets in the dialog are dimmed because the values are inherited from the measured curves (green). The values in the spreadsheets change depending on the lag # entered. By adjusting the lag interval, an excellent fit can often be obtained.

Spreadsheet section

This section includes two separate spreadsheets: Transition Rates and Proportions & Mean Lengths spreadsheets. The Transition Rates spreadsheet contains the rate entries that correspond to the selected option in the Markov Chains section. The Proportions column holds the proportions for each material. The Mean Lengths column contains the average mean length in the vertical direction for each material. The data in these spreadsheets define the Markov chains.

Notes

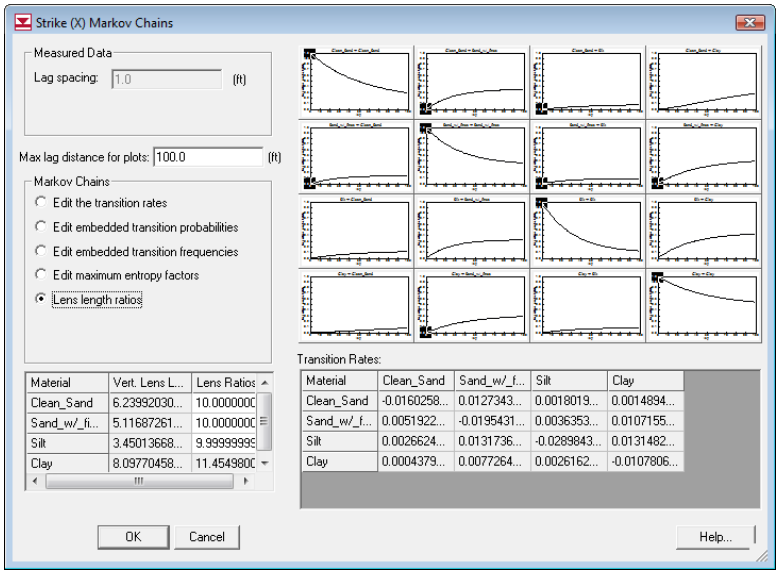
Strike Dip Markov Chain

T-PROGS
Links
T-PROGS
Strike Dip Markov Chain
Vertical Markov Chain
TSIM

Once the vertical Markov chains have been defined, the user is then presented with the *Strike (X) Markov Chains* dialog. This dialog has the same general setup as the vertical dialog.

Strike (X) Markov Chains

There are three minor differences between this dialog and the vertical dialog. The first difference is in the Markov Chains section. The option, Lens width ratios, replaces the Fit curves to a discrete lag option. The Fit curves to a discrete lag option is not applicable because horizontal measured transition probability curves do not exist due to lack of data in the horizontal direction. The Lens width ratios option is the default option and should be used in most cases. This option allows users to apply the transition data entered in the vertical direction to the horizontal direction. The proportion data are directly inherited from the vertical data.



Strike/Dip Markov chain dialog.

The second change is in the Spreadsheet Section. Rather than entering the mean lengths, the user enters a ratio corresponding to the ratio of the lens length in the x direction to the lens length in the z direction. For example, if the lens length for material A is 5 feet in the z direction and the user enters 10.0 for the ratio, then the lens length in the x direction would be 50 feet. The Lens Width Ratios option is particularly useful because the only required input is lens length ratios for the non-background materials. The remaining data are all inherited from the vertical data. The third change is in the Plot Section. Due to the lack of measured data in the horizontal direction, each plot contains only one curve: the Markov chain curve.

Vertical→Lateral Data Conversion

Typical site stratigraphic data is conducive to developing vertical spatial variability. However, rarely is the quantity of data adequate to develop an accurate model in the lateral directions. The combination of Walther's Law and the transition probability approach allows for a logical method of developing lateral spatial variability from vertical spatial variability. Walther's Law states that vertical successions of deposited facies represent the lateral succession of environments of deposition. Therefore, a logical method of generating a 3-D model of spatial variability would be first to develop a 1-D Markov chain in the vertical direction based on site data, assuming there is such data. Second, using Walther's Law and geologic knowledge, one can develop lateral, strike and dip, Markov chains of spatial variability. However, one issue that arises when applying vertical transition trends to lateral directions is how to cope with asymmetric vertical trends like fining upwards. For example, if in the vertical direction, sand tends to deposit on gravel as would be typical in a fluvial deposition, there will be a transition rate associated with the transition of "sand→gravel" and "gravel→sand". The transition of "gravel->sand" will be greater than "sand→gravel" because of the fining upward trend. However, in the lateral direction, which of these transition rates should be applied? Although the trend of sand next to gravel remains in the lateral direction, the transition rates of "sand→gravel" and "gravel→sand" should be equivalent or symmetric as defined by

$$r_{jk} = \frac{p_k}{p_j} r_{kj}$$

The strategy for averaging the vertical transition rates to come up with the lateral rates is given in the three steps below.

1. Compute the lower-half rate (R-ls) that will satisfy symmetry with the upper-half rate (R-u) for the vertical data using the symmetry relation.
2. Set the lower-half rate for the lateral direction (R'-l) equal to (R-l + R-ls)/2
3. Compute the upper-half rate for the lateral direction (R'-u) that will satisfy symmetry with the new lower-half rate (R'-l) using the symmetry relation.

In addition to the averaging technique applied to the off-diagonal terms of the lateral transition rates, an adjustment was made to the diagonal terms of the rate matrix. It will be remembered that the diagonal terms correlate to the average mean lengths by

$$\bar{L}_{j,\phi} = \frac{-1}{r_{jj,\phi}}$$

Generally, in depositional patterns, lateral mean lengths are larger than their counterparts. Therefore, the lateral mean lengths generally need to be increased by a factor F prescribed by the user. This factor F is equal to the ratio of the lateral mean lengths/vertical mean lengths. If the mean length for category K increases by a factor F, the corresponding transition rate will decrease by that same factor F. And the row sum for category K must therefore decrease by the factor F.

Dip (Y) Markov Chains

The dip dialog has an identical appearance and functionality as the strike dialog.

TSIM

T-PROGS
Links
T-PROGS
Strike Dip Markov Chain
Vertical Markov Chain
TSIM

When a T-PROGS simulation has been created, TSIM can be run to generate either material sets on a 3D grid or HUF data for MODFLOW. TSIM is run via the ***TPROGS|Run TSIM*** menu command. This brings up the TSIM Wizard.

Step 1

You can choose to run TSIM now from GMS or save the input files so you can run TSIM later. If you don't run TSIM from GMS, no material sets or HUF arrays will be created.

Step 2

Running TSIM from GMS

Option	Description
Number of realizations	The number of material sets or HUF sets that will be generated.
Seed	The random number seed used by TSIM. This is generated by GMS but can be changed by the user. Running TSIM with the same inputs and see will result in identical outputs.
Max. # of quenching iterations	A TSIM parameter. "No more than five quenching iterations are usually necessary – too many iterations may produce unrealistic artifacts."[1]
Quenching tolerance	A TSIM parameter. "The tolerance limit sets a criteria for terminating quenching based on the value of the objective function as normalized relative to its initial value."
TSIM Output	You can choose to generate material sets or, if a MODFLOW model exists which uses the HUF package, HUF arrays.
Create material probability data sets	This will cause datasets to get created, one per material, where the dataset values equal the probability that that material occurs in the given cell.
Background Grid	The dimensions and number of grid ranks of the background grid are determined from the 3D grid in the project. If generating HUF arrays you can specify the number of grid layers in the vertical (Z) direction.
Target grid layers	Starting at GMS version 8.0 you can target a subset of grid layers. The resulting material sets will have inactive values outside of the targeted area. You can specify the min and max layers or the named layer ranges you wish to target.

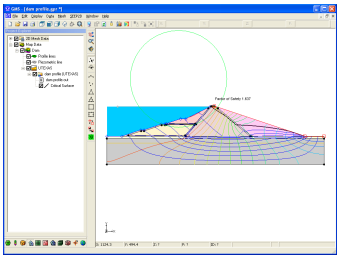
Saving files to run TSIM later

The option in Step 2 if saving the TSIM files for later use are similar to those for running TSIM in GMS. You must specify a file and where it is to be saved. Also, the dimensions and number of ranks of the background grid must be specified. Finally, no material sets or HUF arrays will be generated.

Notes

6.13. UTEXAS

UTEXAS

UTEXAS	
 *UTEXAS Screenshot*	
Model Info	
Model type	Slope stability
Supported versions	4.1.0.3
Developer	Dr. Stephen G. Wright
Documentation	UTEXAS Manual [1] UTEXAS Addendum [2] TexGraph4 Manual [3]
Tutorials	UTEXAS Tutorials

UTEXAS is a slope stability software package created by Dr. Stephen G. Wright of the University of Texas at Austin. UTEXAS is used to analyze slope stability using the limit equilibrium method. The user provides the geometry and shear strength parameters for the slope in question and UTEXAS4 computes a factor of safety against slope failure. The factor of safety for a candidate failure surface is computed as the forces driving failure along the surface divided by the shear resistance of the soils along the surface. UTEXAS4 is a state-of-the-art slope stability code and has been widely used in industry for many years.

GMS versions 6.5 and later include an interface to UTEXAS which allows users to create UTEXAS input files using GMS.

Creating a Model

Existing UTEXAS models can be imported into GMS, or new models can be created from scratch. The UTEXAS interface is entirely within the map module. The model consists of a conceptual model and one or more coverages.

- Importing

  To import an existing UTEXAS model, use the standard ***File|Open*** command. The file to import is the UTEXAS input file. This file may have a *.dat extension which is (unfortunately) the extension UTEXAS expects. GMS creates the UTEXAS conceptual model and coverages from the file being imported.

  Aspects of UTEXAS models can also be imported from a GeoStudio file.

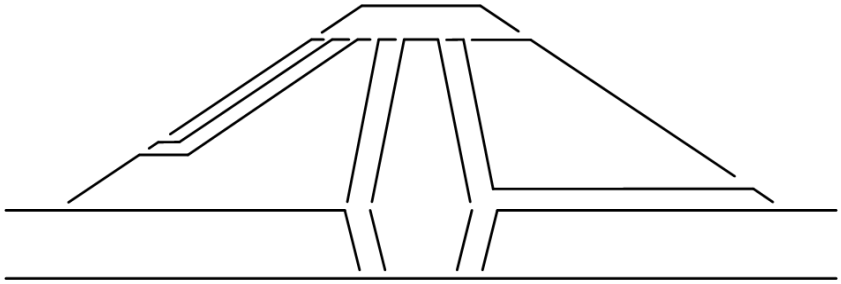
- Creating in GMS

  Creating a UTEXAS model in GMS can be done a variety of ways. The UTEXAS tutorials illustrate these methods.

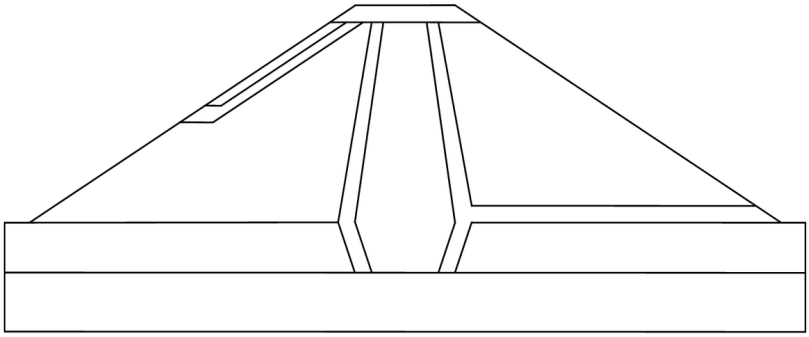
  - Type in the feature point coordinates, then connect the points with arcs

- Create feature arcs with the mouse
- Convert a CAD drawing to feature objects

Profile Lines vs. Arcs and Polygons



Profile lines depicting a dam. Gaps show where lines begin and end (no gaps would exist in an actual model).



Polygons depicting a dam

Profile Lines

UTEXAS uses "profile lines" to define the soil stratigraphy. A profile line is a series of connected points that must go from left to right (or bottom to top) and represent the top of a soil layer. The profile line does not close on itself, and does not cross any other profile lines.

Polygons

GMS does not use profile lines, but instead uses arcs and polygons to define the soil stratigraphy. Using polygons is much easier than using profile lines because there is no need to worry about how to break up the lines such that they all go from left to right (or bottom to top).

When GMS saves the UTEXAS input file, it automatically figures out the profile lines that UTEXAS needs to represent the soil stratigraphy. When importing a UTEXAS file, GMS automatically builds arcs and polygons from the profile lines.

One notable difference between the profile line and polygon approaches is that when using polygons, the bottom-most polygon must be extended downward some distance below the bottom-most profile line to represent the bottom-most soil area.

Material Properties

UTEXAS soil material properties are entered in GMS using GMS materials. Each feature polygon has a material associated with it.

Pore Water Pressure

There are various ways to define pore water pressure. Two of the methods are more advanced: the piezometric line method, and the interpolated pressure method.

Piezometric Line

UTEXAS allows the pore water pressure to be specified using a piezometric line. In GMS, a piezometric line is an arc that is in it's own feature coverage (since the arc will likely overlap arcs in the profile line coverage). The material that uses the piezometric line can specify the coverage the piezometric line (arc) is in.

Interpolated Pressure

Another method for specifying the pore water pressure is "Pressure Interpolated" which uses a set of points where each point has a value representing the pore pressure at that point. In GMS, you can do this by associating a data set from a 2D mesh with the material. The dataset might come from a SEEP2D solution, since SEEP2D is a model in GMS that can solve for pore pressures.

Integration with SEEP2D

SEEP2D is a 2D seepage analysis model that, when combined with UTEXAS, provides a complete and a powerful 2D profile analysis solution. Since these two models are closely related, the conceptual model type for UTEXAS is called "SEEP2D/UTEXAS". One could use the same conceptual model to create both a SEEP2D model and a UTEXAS model. The UTEXAS tutorials illustrate this.

Running UTEXAS

UTEXAS model icon as it appears in the Project Explorer

UTEXAS differs from most models supported by GMS in the way that it is run. Most model codes supported by GMS can be run from inside GMS. Since UTEXAS is a Windows application with no command line options (yet), this is not possible.

Also, GMS typically provides a model wrapper for most of the supported models as a convenient way to watch the progress of the model and automatically read in the results upon completion. Again, since UTEXAS is a Windows application, it is not possible to use a model wrapper.

To run UTEXAS:

- Export the model from GMS
- Launch UTEXAS (outside of GMS)
- Open the exported file in UTEXAS
- Wait for UTEXAS to finish
- Read the solution into GMS

Exporting the Model

To save a UTEXAS model that is in GMS in preparation for running UTEXAS, right-click on the UTEXAS model icon under the conceptual model and select the ***Export*** command. Note that GMS uses the extension *.utx instead of *.dat when saving the UTEXAS input file.

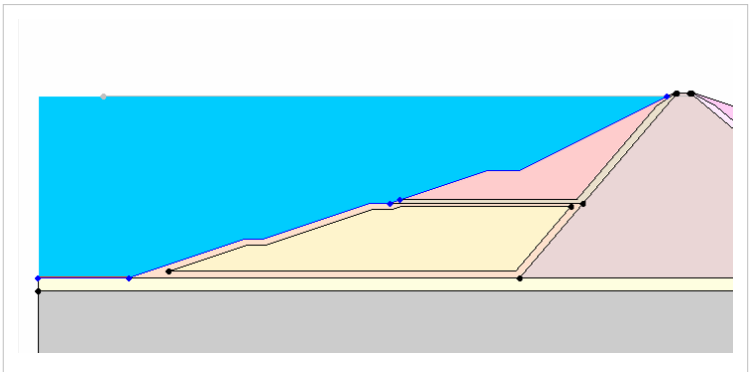
Reading the Solution

To read the solution generated by UTEXAS into GMS, right-click on the UTEXAS model icon under the conceptual model and select the ***Read Solution*** command. The UTEXAS solution will have an extension of *.out.

Display Options

A number of display options are associated with UTEXAS. To find these, select the **Display Options** button, click on the *Map Data* item from the list, and select the *UTEXAS* tab on the right.

UTEXAS Display Options

Option	Description
Distributed loads	Arcs that are designated as distributed loads. Arrows are drawn at the spacing and scale specified to show the loading.
Head	 Arcs designated as Head arcs have the space above them filled with the specified color.
Starting circle	The starting circle specified in the *UTEXAS Analysis Options* dialog.
Factor of safety	If a solution exists, the factor of safety.
Critical surface	If a solution exists, the critical surface.
Limiting depth	The limiting depth specified in the *UTEXAS Analysis Options* dialog.
Tangent line	The horizontal tangent line specified in the *UTEXAS Analysis Options* dialog.

Unsupported Features

Not all of the features of UTEXAS are currently supported in the GMS interface. Support for more features will be added in future versions of GMS. The following features are not supported.

Command Words

- LABel
- NO compute
- OTHer units
- SIUnits /or/ SI
- UT3
- UT4
- '***'

Group B, Profile Lines

- Table 6.2, Group B - Profile Line Data Input Format - Import Mode

Group C, Material Properties

- Table 7.1, Group C - Material Property Data Input Format
  - Line 4
    - Reference (R)
    - Anisotropic shear (A)
    - Interpolate Strengths (I S) (version 7.0 and earlier. Supported starting at GMS version 7.1)
  - Line 6
    - Interpolate Ru values (I R)

Group D, Piezometric Line

- Table 8.1, Group D - Piezometric Line Data Input Format
  - Line 2
    - Field 2, Unit weight of water

Group E, Interpolation Data

- Table 9.1 - GMS will export this table but cannot import it
- Table 9.2 - GMS won't import or export this table

Group G, Distributed Loads

- Table 11.1, Group G - Distributed Load Data Input Format for Individual Points
  - Line 2
    - Field 4, Shear stress

Group K, Analysis/Computation

- Table 14.2a
  - Line 3
    - Field 1, "T" (or "TANGENT") (version 7.0 and earlier. Supported starting at GMS version 7.1)
- Table 14.2c, "Fixed" Grid
  - Line 3
    - Field 1, "T" (or "TANGENT") (version 7.0 and earlier. Supported starting at GMS version 7.1)
- Table 14.2d, "Fixed" Grid
  - Not supported in version 7.0 and earlier. Supported starting at GMS version 7.1. Also, although multiple criteria can be used to specify the radius, GMS only supports using one.
- Table 14.3, Summary of Sub-Command Words for Group K Data
  - 1 (Same as "SIN" which GMS does support)
  - 2 (Same as "TWO" which GMS does support)
  - 3 (Same as "THR" which GMS does support)
  - ARC (Arc length)
  - BAS (Base length)
  - CHA (Change)
  - CRI (Critical shear surface)
  - FAC (Factor of Safety)
  - FOR (Force imbalance)
  - INC (Increments for subdivision)
  - ITE (Iteration)
  - MIN (Minimum weight)
  - MOM (Moment imbalance)
  - PAS (Passes)
  - RES (Restrictions)
  - SAV (Save "n" most)
  - SEI (Seismic coefficient) (version 7.0 and earlier. Supported starting at GMS version 7.1)
  - SID (Side force inclination)
  - SOR (Sort radii)
  - STO (Stop)
  - SUB (Subtended angle)
  - TRI (Trial)
  - UNI (Unit weight of water)
  - UNS (Unsort)
- %206.5/models/UTEXAS4%20Manual.pdf UTEXAS4 Manual [4]
- %206.5/models/UTEXAS4%20Addendum.pdf UTEXAS4 Addendum [4]
- Dr. Stephen G. Wright [5]
- UTEXAS article on Wikipedia [6]

- Jun 1987 CETN-III-34 Checking Stability of Bulkheads Located in Ports, Harbors and Coastal Waterways Against Shear Failure Computer Program: UTEXAS2 [7]

References

[1] http://gmsdocs.aquaveo.com/UTEXAS4%20Manual.pdf
[2] http://gmsdocs.aquaveo.com/UTEXAS4%20Addendum4.pdf
[3] http://gmsdocs.aquaveo.com/TexGraf4%20Manual.pdf
[4] ftp://pubftp.ems-i.com/download/GMS/GMS
[5] http://www.ce.utexas.edu/prof/wright/home.html
[6] http://en.wikipedia.org/wiki/UTEXAS
[7] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/cetn-iii-34.pdf

UTEXAS Display Options

The properties of all UTEXAS properties that GMS displays on the screen can be controlled through the UTEXAS tab of the *Display Options* dialog. This dialog is opened by right clicking on the Map Data entry in the Project Explorer, selecting the ***Display Options*** command, and then selecting the UTEXAS tab. This tab is only visible when there is a UTEXAS simulation. It can also be accessed from the from the *Display* menu or the **Display Options** macro. The following table describes the display options available for the UTEXAS model.

Display Option	Description
Distributed loads	Controls the display of arrows representing distributed loads placed on an embankment or dam. The line color can be adjusted by clicking on the button to the right of the toggle. If the **Auto X spacing** is toggled on, the arrows will automatically be distributed along the length of the distributed load. However, the user can specify the X spacing by turning off and the **Auto X spacing** toggle and specifying an X spacing the edit field. This value represents a real world value. The **Auto Y scale** controls the length of the arrow tails representing the distributed load. If the **Auto Y spacing** toggle is turned off, the arrow tails will be drawn using the Y scale specified in the edit field.
Head	Controls the display of the head fill color. The head will be displayed as filled polygon representing the amount of head that is acting on an embankment. This color can be adjusted by clicking on the button to the right of the toggle.
Starting circle	Controls the display of the line representing the starting circle to be used by UTEXAS in performing the slope stability analysis. The line type, line width, and line color can be changed by clicking the on button to the right of the toggle.
Factor of safety	Controls the display of the factor of safety computed by the UTEXAS simulation. The font type, font size, and font color can be adjusted by clicking the on button to the right of the toggle.
Critical surface	Controls the display of the line representing the critical surface determined by the UTEXAS simulation. The line type, line width, and line color can be adjusted by clicking the on button to the right of the toggle.
Limiting depth	Controls the display of the line representing the limiting depth to be used in the UTEXAS simulation. The line type, line width, and line color can be changed by clicking the on button to the right of the toggle.

6.14. WASH123D

WASH123D Display Options

The properties of all WASH123D boundary conditions that GMS displays on the screen can be controlled through the WASH123D tab of the *Display Options* dialog. This dialog is opened by right clicking on the 3D Mesh Data entry in the Project Explorer, selecting the ***Display Options*** command, and then selecting the WASH123D tab. This tab is only visible when there is a FEMWATER simulation. It can also be accessed from the from the *Display* menu or the **Display Options** macro. The following table describes the display options available for the WASH123D model.

Display Option	Description
Well	Controls the display of well flow boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Well (super pump node)	Controls the display of well super pump nodes flow boundary conditions. The fill color can be adjusted using the button to the left of the display toggle. The radius of the well can also be adjusted in the edit field below the Well (super pump node) toggle.
Head	Controls the display of head flow boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Equivalent head	Controls the display of equivalent head flow boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Spec. flux	Controls the display of specified flux flow boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Variable flux	Controls the display of variable flux flow boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Enforced total head	Controls the display of enforced total head flow boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Init. enforced total head	Controls the display of initial enforced total head flow boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Discharge x-sect.	Controls the display of discharge cross section flow boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Flux	Controls the display of flux boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Well conc.	Controls the display of well concentration transport boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Spec. conc.	Controls the display of specified concentration transport boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Spec. mass flux	Controls the display of specified mass flux transport boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
Variable conc.	Controls the display of variable concentration transport boundary conditions. The symbol, symbol size, and symbol color can be adjusted using the button to the left of the display toggle.
BC values	Controls the display of boundary condition values. The font, font size, and font color can be adjusted using the button to the left of the display toggle.
Display symbol legend	Controls the display of a legend containing the symbols for the boundary conditions used in the FEMWATER simulation.

Check All	By clicking this button, all of the display options are turned on.
Check None	By clicking this button, all of the display options are turned off.

7. Modeling

7.1. Calibration

Model Calibration

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

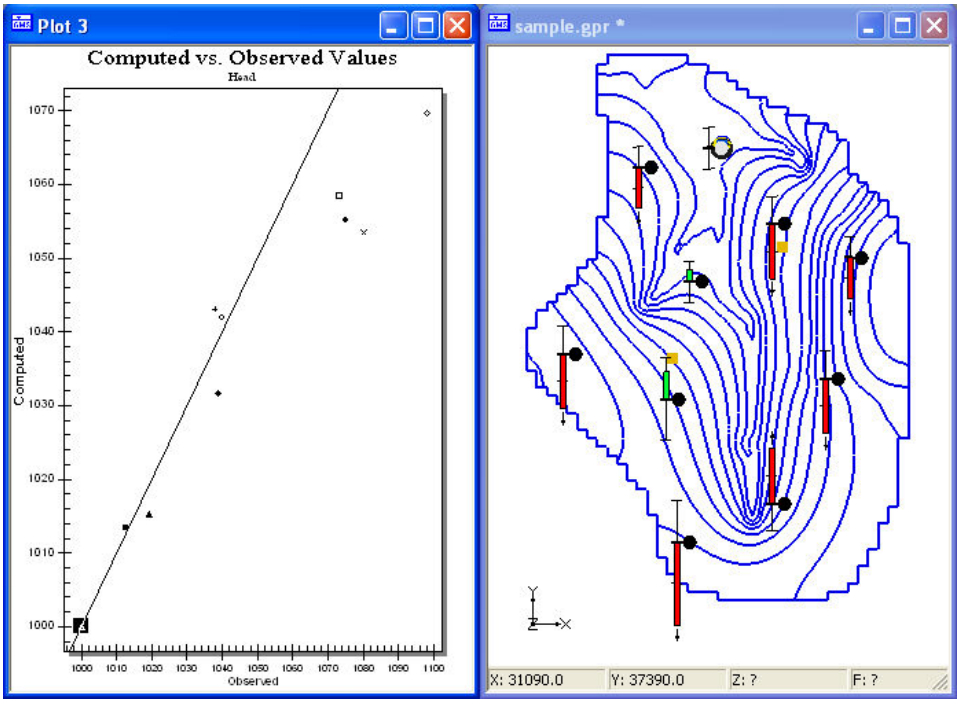
Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

Calibration is the process of modifying the input parameters to a groundwater model until the output from the model matches an observed set of data. GMS includes a suite of tools to assist in the process of calibrating a groundwater model. Both point and flux observations are supported. When a computed solution is imported to GMS, the point and flux residual errors are plotted on a set of calibration targets and a variety of plots can be generated showing overall calibration statistics. Most of the calibration tools can be used with any of the models in GMS. Automated Parameter Estimation is supported for MODFLOW models via PEST.

Two types of observations can be defined in GMS: point observations and flow observations. Both types of observations are defined in the map module and are associated with points, arcs, and polygons. Point observations represent locations in the field where some value has been observed. In most cases, the points will correspond to observation wells and the value will be the elevation of the groundwater table (the head). Flow observations represent linear or areal objects such as streams and reservoirs where the gain or loss between the aquifer and the object has been measured or estimated. Both point and flow observations can be assigned a confidence interval or calibration target. While point observations can be used with any model, flow observations can only be used with MODFLOW.

Once a set of observed point and flow values has been entered, each time a model solution is imported, GMS automatically interpolates the computed solution to the observation points. For some models (MODFLOW) the model outputs the computed values at the observation points. Also, MODFLOW outputs the computed flow for the flow observations. A calibration target representing the magnitude of the residual error is displayed next to each observation point and each flow object as shown below. The size of the target is based on the confidence interval or

the standard deviation. In addition to the calibration targets next to the observation points, you can choose to display any of a number of statistical plots.



Calibration Targets and Plots of Calibration Statistics.

Automated Parameter Estimation

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR

Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

One of the tools provided in GMS for model calibration is automated parameter estimation. With automated parameter estimation, an external utility, sometimes called an "inverse model", is used to iteratively adjust a set of parameters and repeatedly launch the model until the computed output matches field-observed values. Parameter estimation is used in conjunction with the point observations and the flow observations.

Automated parameter estimation is supported in GMS for the MODFLOW simulations using PEST a general purpose parameter estimation utility developed by John Doherty of Watermark Computing.

Inverse models should only be used carefully and with a full understanding of the assumptions, equations, and methods involved. It is suggested that the user read the available documentation on the inverse model being used. Only the steps involved in setting up an inverse model are described in this document.

Basic Steps

The basic steps involved in using an inverse model for parameter estimation are follows:

1. Create a Working MODFLOW Model

The first step is to create your MODFLOW model and run a simulation. Before launching the inverse model, you need to have a MODFLOW model that successfully converges and you need to determine a good set of starting values for your parameters. Once you have a solution it is also a good idea to copy the computed heads from your solution to your starting heads array. This ensures that as the inverse model modifies the parameters and runs MODFLOW repeatedly, it is more likely that MODFLOW will quickly converge each time it is launched.

2. Enter the Observations

Once you have a working MODFLOW model, you should enter your head and flux observations. Head observations are entered as points using an observation coverage in the Map module. Flow observations are assigned directly to arcs and polygons in source/sink coverages. Each of the observations is assigned a weight that is saved to the inverse input files.

3. Turn on the Inverse Model

You must select an inverse model. Bring up the Global Options dialog and select either the Parameter Estimation or Stochastic Inverse Model button depending on whether a stochastic simulation is being run.

4. Parameterize the model

The next step is to parameterize your model. See the Parameters page for more details.

5. Create a Parameter List

The next step is to create the parameter list. See the Parameters page for details.

6. Set parameter estimation options

Once the parameter list is set up, you may wish to edit the general Parameter Estimation options. These options include the output control and convergence criteria.

7. Edit the Group Weight Multipliers

The group weight multipliers can be edited to adjust the relative weight of the head and flux observations.

8. Edit the PEST ASP Package

Edit the MODFLOW PEST ASP Package if necessary in order to ensure a stable solution.

9. Save and Run MODFLOW Model

Once all of the inverse model options have been set, the next step is to save the MODFLOW model using the ***Save/Save As*** command in the *File* menu. Next, run MODFLOW and the inverse model will run with MODFLOW. The inverse model will then be launched in a separate window or the model wrapper in which you will see information relating to the MODFLOW runs and the status of the objective function. Depending on the problem, the inverse model may take anywhere from several minutes to several hours (or days) to run to completion. When the inverse process is completed successfully, GMS automatically launches a MODFLOW forward run with the optimal values computed by the inverse model. Thus, the solution will reflect the optimal values computed by the inverse

model.

10. Viewing the Optimal values

When the inverse model is finished, it writes out a text file containing the set of parameter values corresponding to the minimum calibration error. These values can be viewed with the Import Optimal Values button. This copies the optimal parameter values to the Starting Value field in the Parameter List.

Sensitivity Analysis

At each PEST iteration, PEST computes the sensitivities of each of the parameters. This information is available in the "*model*.sen" file (where *model* is the name of the MODFLOW model). PEST records the composite sensitivity and the relative composite sensitivity of each parameter in this file. This information is useful in determining which parameters have the greatest effect on the model as well as which parameters have the least effect on the model. Thus, the "insensitive" parameters can be removed or held constant in a subsequent PEST run.

For a more detailed description of parameter sensitivity see section 5.3.2 of the PEST manual.

Parameter Estimation Dialog

Options affecting parameter estimation can be changed via the Parameter Estimation Dialog.

Parameter Estimation Dialog

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR

Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

The Parameter Estimation dialog allows the editing of inverse modeling options that are specific to PEST. This dialog is available when Parameter Estimation is chosen as the Run option in the *Global Options\Basic Package* dialog.

Max number of iterations (NOPTMAX)

This value (NOPTMAX) sets the maximum number of optimisation iterations that PEST is permitted to undertake on a particular parameter estimation run. If you want to ensure that PEST termination is triggered by other criteria, more indicative of parameter convergence to an optimal set or of the futility of further processing, you should set this variable very high. A value of 20 to 30 is often appropriate.

If NOPTMAX is set to 0, PEST will not calculate the Jacobian matrix. Instead it will terminate execution after just one model run. This setting can thus be used when you wish to calculate the objective function corresponding to a particular parameter set and/or to inspect observation residuals corresponding to that parameter set.

If NOPTMAX is set to −1, PEST will terminate execution immediately after it has calculated the Jacobian matrix for the first time. The parameter covariance, correlation coefficient and eigenvector matrices will be written to the run record file, and parameter sensitivities will be written to the sensitivity file; these are based on the initial parameter set supplied in the PEST control file.

Max number of iterations with no improvement (NPHINORED)

If PEST has failed to lower the objective function over NPHINORED successive iteration it will terminate execution. A value of 3 or 4 is often suitable for this variable.

Advanced Options

Max number of relative convergence iterations (NPHISTP); Relative convergence limit (PHIREDSTP)

If, in the course of the parameter estimation process, there have been NPHISTP optimisation iterations for which

$(\Phi i - \Phi min)/\Phi i \leq$ PHIREDSTP (4.6)

(Φi being the objective function value at the end of the i'th optimisation iteration and Φmin being the lowest objective function achieved to date), PEST will consider that the optimisation process is at an end.

For many cases 4 and 0.01 are suitable values for NPHISTP and PHIREDSTP respectively. However you must be careful not to set NPHISTP too low if the optimal values for some parameters are near or at their upper or lower bounds. In this case it is possible that the magnitude of the parameter upgrade vector may be curtailed over one or a number of optimisation iterations to ensure that no parameter value overshoots its bound. The result may be smaller reductions in the objective function than would otherwise occur.

Max number of relative parameter change iterations (NRELPAR); Relative parameter change criterion (RELPARSTP)

If the magnitude of the maximum relative parameter change between optimisation iterations is less than the Relative Parameter Change Criterion (RELPARSTP) over Max # of Relative Parameter Change Iterations (NRELPAR) successive iterations, PEST will cease execution.

All adjustable parameters, whether they are relative-limited or factor-limited, are involved in the calculation of the maximum relative parameter change. RELPARSTP is a real variable for which a value of 0.01 is often suitable. NRELPAR is an integer variable; a value of 2 or 3 is normally satisfactory.

Max relative parameter change per iteration This (RELPARMAX) is the maximum relative change that a parameter is allowed to undergo between optimisation iterations.

Max factor parameter change per iteration (FACPARMAX)

This value is the maximum factor change that a parameter is allowed to undergo during an iteration.

Run-time matrix options

Select these options if you want these arrays written to external files. These arrays will also be written to the PEST output file (*.rec).

Measurement obj func upper limit (PHIMLIM)

This is the upper limit of the measurement objective function (i.e., the upper level of model-to-measurement misfit) that is tolerable when trying to minimise the regularisation objective function Φr.

Measurement obj func accept. level (PHIMACCEPT)

This is the acceptable level for the measurement objective function that PEST uses to change its method of calculating the Marquardt Lamdas (see PEST documentation).

Normally PHIMACCEPT should be about 5% to 10% greater than PHIMLIM. However if PEST is performing well, you may wish to make it closer to PHIMLIM than this. In choosing the best parameter set at any stage of the optimisation process (for recording in the parameter value file) PEST looks at all parameter sets for which it has carried out model runs up to that point in the process. If any of these runs have resulted in an objective function less than PHIMACCEPT, it then searches from among these runs for the parameter set which gave rise to the lowest regularisation objective function. If PHIMACCEPT is set too close to PHIMLIM, PEST's selection of the best parameter set may be restricted somewhat, for there may be some parameter sets for which the measurement objective function Φm is just above PHIMACCEPT but for which Φr is quite low. Alternatively, if PHIMACCEPT is set too large, then PEST might not try hard enough to reduce Φm to Φmi, preferring instead to work within the weaker constraint set by PHIMACCEPT. When working in regularisation mode, PEST prints out Φr and Φm for every parameter upgrade attempt. It will be apparent from this information whether PHIMACCEPT has been set correctly.

FRACPHIM

PEST ignores the value supplied for FRACPHIM unless it is greater than zero. A value of between zero and 1.0 (but normally less than about 0.3) can be supplied for this variable if you are unsure what value to use for PHIMLIM. See the PEST documentation.

Automatic user intervention (AUI)

When the optimization process undertaken by PEST appears to be going nowhere, the situation can often be remedied by selectively withdrawing certain parameters (normally the most insensitive ones) from the parameter estimation process. This process has been automated in PEST using the AUI option. For more information on AUI see the PEST documentation (section 5.7 of the PEST manual).

It should be noted that use of the SVD process in PEST is usually more effective than AUI as the SVD process removes insensitive parameters.

Parallel PEST

GMS allows you to run PEST in parallel on a single machine to take advantage of multiple cores. This can greatly speed up the PEST runs.

- Number of slaves

  "Slaves" refers to the separate processes that will be run in parallel. You can specify the number of slaves or, if not specified, GMS will automatically determine the number of cores available on the machine and use that for the number of slaves.
- Wait time

  The wait time is the amount of time that PEST will pause at certain strategic places in their communication. Normally the default value should work fine. However, if either PEST reports a sharing violation on your hard drive then you should increase the value of the wait time.

SVD (Singular Value Decomposition)

The *Use SVD* toggle is available to turn on the SVD functionality in PEST. For a more complete explanation of SVD, see the PEST manual [1] in section 8.4. The inputs to the SVD process can be edited by selecting the **SVD Options** button.

The SVD process analyzes the parameters that are currently part of the parameter estimation process and removes parameters that are not helping to solve the problem. The user can limit the number of parameters used by specifying a maximum number of singular values (MAXSING) or by specifying EIGTHRESH. By default MAXSING is set to 1000 and EIGTRESH to 1E-7, so that the number of parameters will be limited by the value of EIGTHRESH.

- MAXSING – maximum number of singular values to include in the inversion process.
- EIGTHRESH – a ratio of the lowest to highest parameter eigen value (a value of 1E-7 is usually sufficient).
- EIGWRITE – controls output written to the *.svd file by PEST. When the value is 1 a more verbose file is written that can become quite large. By default the value of EIGWRITE is set to 0.

SVD-Assist

The **Use SVD-Assist** toggle turns on the SVD-Assist process for PEST. This process is particularly advantageous for models that have hundreds or thousands of parameters (such as pilot points). SVD-Assist involves 3 basic steps.

1. First, PEST runs MODFLOW once for every parameter in order to compute a matrix. This information is used to create super parameters that are combinations of the parameters originally specified.
2. Second, SVDAPREP is run to create a new PEST control file. The options for SVDAPREP are entered by selecting the **SVD-Assist Options** button. For more information on each of these options see the PEST manual in section 8.5.4.2. The most important option entered is the *Specify # super param* and this is set to **No** by default. When this option is set to No, then the information written to the *.svd file will be used to specify the number of super parameters.
3. Third, PEST runs using the new control file written by SVDAPREP. This should result in significantly fewer model runs for each PEST iteration. This often results in an order of magnitude reduction in the number of runs required for each PEST iteration.

Tikhonov Regularization

Tikhonov Regularization (prior information for pilot points). This section of the dialog allows the user to select the method for include Tikhonov regularization in the PEST run. If neither option is turned on then regularization will not be included in the PEST run.

The first option, Preferred homogeneous regularization, is the option that GMS has always supported with pilot points. When using this option, prior information equations are included that impose a homogeneity constraint on the pilot points. This means that in absence of other information, pilot points that are near to one another should have about the same value.

- Prior information power factor

  This is used to change the weight applied to the prior information equations for the pilot points. A value of 1 is normally sufficient. The prior information equations impose a homogeneity constraint on the pilot points. This means that in absence of other information, pilot points that are near to one another should have about the same value. When the prior information equations are created GMS will compute an inverse distance weight between each pilot point and all other pilot points for a given parameter. This weight is then raised to the power of the *Prior information power factor* and assigned to the equation for a given pair of points. So to increase the homogeneity constraint (assign a higher weight to the prior information equation), the user should decrease this value. To decrease the homogeneity constraint (assign a lower weight to the prior information equation), the user should increase this value.

The second option, Preferred value regularization, is a new option beginning with GMS version 8.0. When this option is used, prior information equations are included that constrain the pilot points near their starting values. Using this option the user can give a set of starting values to the pilot points based on field data or professional judgment. PEST will only change those values if necessary to calibrate the model.

Also new in GMS 8.0, parameters of different types (HK, RCH) are put into different regularization groups. According to John Doherty, this helps differentiate weighting amongst pertinent prior information equations.

References

[1] http://gmsdocs.aquaveo.com/pest.pdf

PEST

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

PEST is a general purpose parameter estimation utility developed by John Doherty of Watermark Computing. The PEST interface in GMS can be used to perform automated parameter estimation for MODFLOW. The steps involved in setting up a PEST run are described in the automated parameter estimation topic.

The PEST options are edited by selecting the ***Parameter Estimation*** command in the *MODFLOW* menu and then clicking on the PEST tab.

PEST version support

The following table shows the version of PEST supported by different versions of GMS. To see the latest released version of PEST visit http://www.pesthomepage.org/Downloads.php.

GMS	PEST
GMS 6.5	PEST 10.0
GMS 7.0	PEST 10.0
GMS 7.1	PEST 12.0
GMS 8.0, 8.1, 8.2	PEST 12.1
GMS 8.3, 9.0	PEST 12.2
GMS 9.1	PEST 12.3

Run Options

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Once the MODFLOW simulation has been initialized, the next step is to enter the data required by the Global Options/Basic package. This includes data defining fundamental program options such as the computational time intervals (stress periods), an array defining which cells are inactive and which cells have constant heads, an array of starting head values for a transient simulation, and a set of flags defining which of the other packages are to be used. The input data for this package should be entered before editing any of the other packages. The MODFLOW *Global Options\Basic Package* dialog is accessed through the ***Global Options*** command in the *MODFLOW* menu. The options in the dialog are as follows:

Headings

The two headings are optional text strings which are written to the MODFLOW text output file.

Run Options

The run options are not part of the actual MODFLOW input files, but present different ways that MODFLOW 2000 can be run from GMS.

Forward Run

A forward run is a normal MODFLOW simulation. If parameters have been defined for either a stochastic or parameter estimation run, the mean or starting values of the parameters are used for the forward run.

Sensitivity Analysis

The sensitivity analysis option is used to perform a sensitivity analysis on a set of selected parameters. The results can be presented with a set of plots or by contouring data sets representing grid sensitivities. This option is only available with the LPF and HUF flow packages. This option is not available in GMS 6.5 and later (see MODFLOW with HDF5).

After running PEST, sensitivities are displayed in the .sen file. Click here for more information.

Parameter Estimation

In the parameter estimation mode, a set of parameters is found that minimizes the residual between observed and simulated heads and flows. These options are described in more detail on the Automated Parameter Estimation page. This option is only available with the LPF and HUF flow packages.

Stochastic Simulation

The stochastic simulation option is used to perform a stochastic simulation using the Monte Carlo or Latin Hypercube methods for randomizing a selected set of parameters. Stochastic simulations can also be performed using a material set approach (indicator simulations). A risk analysis wizard can be used to analyze the results for a probabilistic threshold analysis or a probabilistic capture zone delineation. These options are described in more detail on the stochastic simulation page.

Stochastic Inverse Model

If the material set option is used for stochastic simulations, the "stochastic inverse" option can be used to perform automated parameter estimation on each of the N candidate model instances in the stochastic. The weighted RMS errors for each of the resulting optimized solutions can be used to weight the computations performed by the risk analysis wizard. This option is only available with the LPF and HUF flow packages.

Model Type

The simulation can be designated as either steady state or transient. If a steady state simulation is specified, certain portions of the MODFLOW interface such as the *Stress Period* dialog are inactivated since they are not relevant.

No Flow Head

This is the value of head to be assigned to all inactive (no flow) cells (IBOUND=0) throughout the simulation.

Stress Periods

The Stress Periods button is used to bring up the *Stress Period* dialog. The stress periods should be defined before any of the sources/sinks are entered. If the steady state option is selected, the **Stress Periods** button is dimmed.

Packages

The Packages button brings up the *Packages* dialog.

Units

The Units button the *Basic Package* dialog brings up the *Units* dialog.

IBOUND

Values

The IBOUND button of the *Global Options\Basic Package* dialog brings up the array editor. The IBOUND array contains a value for each cell in the grid defining the type of the cell as constant head, inactive, or variable head.

Constant Head

A negative value indicates that the cell has a constant head. The value of the constant head is defined in the starting heads array.

Inactive

An IBOUND value of zero indicates that the cell is inactive (no-flow).

Variable Head

A positive IBOUND value indicates that the cell has a variable head (i.e., the head value will be computed as part of the simulation).

Editing the IBOUND Array

There are several ways to change the active/inactive status (positive vs. zero) of a cell before or after initializing the IBOUND array. One method is to directly edit the IBOUND array using the *IBOUND* dialog. Another method is to select the ***Cell Attributes*** command in the *MODFLOW* menu. In most cases, the most efficient method is to use the ***Activate Cells in Coverage*** command in the Map module. This method uses a polygon to define the active and inactive regions.

The constant head cells are typically assigned or edited in one of three ways. One method is to directly edit the IBOUND array. Another method is to select a set of cells and use the *Cell Properties* dialog in the *MODFLOW* menu. The simplest method is to define the constant head zones using feature objects as part of a conceptual model in the Map module.

Starting Heads

The starting head values are used as initial conditions for head for both steady state and transient simulations. The **Starting Heads** button on the left side of the *Basic Package* dialog is used to enter the values of the starting heads array. Selecting the **Starting Heads** button brings up the MODFLOW Array Editor.

Top Elevation

The Top Elevation button brings up an array editor that allows you to edit the top elevation of each layer. GMS requires that the top and bottom elevations for adjacent layers be the same. This means that when you edit the bottom of one layer, move down to the next layer, and edit the top of that layer, the bottom of the previous layer will also be changed. Note that the top and bottom layer elevations must be entered regardless of the which layer data package (BCF, LPF, HUF) is chosen even though these values may not be used because of certain package options.

Bottom Elevation

The Bottom Elevation button brings up an array editor that allows you to edit the bottom elevation of each layer. Note that the top and bottom layer elevations must be entered regardless of the which layer data package (BCF, LPF, HUF) is chosen even though these values may not be used because of certain package options.

Confining Beds

The Layer Confining Beds dialog, accessed from the *Global Options/Basic Package* dialog, allows you to select which layers will have a confining bed below them. The last layer in the grid may not have a confining bed.

Confining beds are most common in older models where the user was trying to conserve memory in solving the groundwater model. In general, it is recommended that you explicitly model all of the layers in the system instead of using confining beds. This feature is supported in GMS to make it possible for users to read in older models.

In the spreadsheet, the user may turn on the confining bed option for any of the layers except the last layer in the grid.

The Confining Bed Elevations button brings up the array editor that allows the user to edit the elevation of each layer.

Observations

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

Point Observations

The primary type of field data used in a typical calibration exercise is point observations. Point observations represent values that are measured at some location in the field. Point observations generally correspond to water table elevations measured at observation wells. However, multiple observed values can be defined at each observation point. Observation points are managed in the Map module using the *Coverage Setup* dialog.

Flow Observations

Flow observations represent gains or losses between aquifers and streams or reservoirs. In addition to point observations, flow observations are an essential part of a calibration exercise for a flow model. If calibration is attempted using point observations only, there may be many combinations of parameters such as hydraulic conductivity and recharge that will result in the same head distribution. Adding one or more flow observations serves to "pin down" the flow quantity resulting in a set of hydraulic conductivities and recharge values that are more likely to be unique.

While the point observation tools are model independent, GMS only supports flow observations for MODFLOW and FEMWATER. With MODFLOW, observed flows are assigned to selected arcs and polygons making up the

MODFLOW conceptual model in the Map module. When a MODFLOW solution is imported, the computed flows are read for the arcs and polygons and compared with observed values.

With a FEMWATER simulation, observed values cannot be assigned to objects in the FEMWATER conceptual model. However, when a FEMWATER solution is imported, the computed flows on selected model boundaries can be automatically summed. Comparison of computed vs. observed flows must then be made manually.

Observation Weights

When performing automated parameter estimation, a set of head and flow observations are defined using points, arcs, and polygons in the Map module. When entering the point and flow observations, care should be taken when entering the calibration interval and confidence values. These values are used to determine the weights assigned to each observation in the inverse model. The weight is multiplied by the residual for the observation in the objective function. The weight that is sent to the inverse model input files by GMS is computed as

$$\frac{1}{(\text{Standard Deviation})^2 * \text{Group Weight}}$$

Note that GMS will automatically convert from an interval and a confidence to a standard deviation or you can directly enter the standard deviation.

Group Weight Multipliers

In addition to the individual weight, a group weight can also be assigned. Group weights are assigned using the Group Weight section of the *Observations* dialog. This dialog is accessed by selecting the ***Observations*** command in the *MODFLOW* menu.

A group weight can be assigned to each of the following observation types:

1. **Head observations.**
2. **Constant head flow observations.**
3. **River flow observations.**
4. **General head flow observations.**
5. **Drain flow observations.**
6. **Stream flow observations.**

Options 2-6 correspond to flow observations that are defined using the Observed flow rate option.

The default value for the group weights is 1.0. The default value can be changed to give a larger influence to a particular observation type. For example, if a particular model had sixteen head observations and one flux observation corresponding to a stream gage, a better solution may be obtained by increasing the flux group weight to give more weight to the stream gage measurement.

Observations Dialog

This dialog is used to manage which coverages that contain MODFLOW observation data will be applied to the current simulation. It is also used to apply group weights. You can only access this dialog when observations have been created using the Map Module in a MODFLOW conceptual model.

Group Weights

These weights can be used to emphasize (or deemphasize) a type of observation for the simulation.

Coverages

This spreadsheet allows you to choose which observation coverages will be used in the current simulation. This can be helpful if you have observations for a site from different times, but you only want to use one of the times.

Export Trans. Obs.

Beginning with version 8.0, this button allows you to export a *.csv (comma separated values) file with transient observation data. This file can be loaded into excel to create plots of the transient observation values vs. the model computed values. An example of using this data is included in the MODFLOW- Managing Transient Data tutorial.

Confidence Interval and Standard Deviation

The interval and standard deviation are related by the following equation:

$$sd = \frac{CI}{z}$$

where sd is the standard deviation, CI is the confidence interval, and z is the the "z statistic" based on the specified confidence and the normal distribution. The user can enter an interval and confidence, and the standard deviation will be computed; or the user can enter a standard deviation and confidence, and the interval will be computed.

In practical terms entering an interval (or standard deviation) and confidence is an indication of how much error the user believes is associated with the observed value.

The standard deviation becomes important when using an inverse model. The weight assigned to each of the observations points is a function of the standard deviation. This weight is used in the objective function that the inverse model tries to minimize. An observation point with a small standard deviation will have a greater influence on the objective function than a point with a large standard deviation.

The user must enter an interval (or standard deviation) and confidence in the *Properties* Dialog. Although these values are rarely quantified the following examples may prove helpful.

If the user had the following set of head measurements for one observation well:

Date	Head
May 10, 1998	55.0
July 10, 1998	50.5
September 5, 1998	48.6
November 15, 1998	49.1
February 2, 1999	50.8
March 12, 1999	54.0
April 1, 1999	57.2

The mean of the data is 52.17 and the standard deviation is 3.25. The user could enter this standard deviation and a confidence of 95%.

Many times the user does not have this much data available. Usually the user will only have one measured value. In this case the user must use engineering judgement to estimate an interval or standard deviation. For example, if I have a single head measurement of 45.7 ft and I believe that observation to be accurate to within a 1 foot. Then I would enter 1 foot as my interval.

Plot Wizard

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

Plots are useful for many purposes, such as extracting data from two or three dimensional objects and model verification. Plots are created through the Plot Wizard.

Plot Wizard Steps

The Plot Wizard (***Display*** | ***Plot Wizard***) is used to create 2D plots. The plot wizard is composed of two steps described below. In addition, the types of plots that can be created are described and illustrated below.

Step 1

In the first step, the plot type is selected. The types include:

- Computed vs. Observed Data
- Computed vs. Observed Data (Weighted)
- Residual vs. Observed Data
- Residual vs. Observed Data (Weighted)
- Parameter Sensitivities
- Error vs. Simulation
- Error vs. Time Step

- Error Summary
- Time Series
- Active Data Set Time Series Plot
- S/S Flow vs. Time
- Flow Budget vs. Time
- Gage Package Value vs. Time

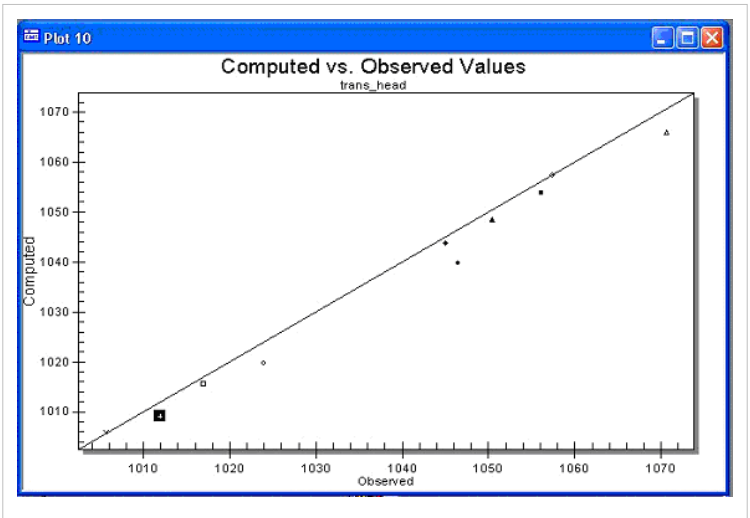
A sample and explanation are displayed for each plot type. The **Next >** button is undimmed if the necessary data for the selected plot type exists in the current project. If the **Next >** button is disabled, a help text explaining the problem is displayed below the Plot Type box.

Step 2

In the second step, the attributes of each plot are set. The attributes associated with each plot type are explained below and can be located quickly by clicking on the desired plot type listed above. The options depend on the plot type and will be described below.

Computed vs. Observed

A Computed vs. Observed plot is used to display how well the entire set of observed values match a model solution. A 45^{o} line is drawn on this plot, which represents a perfect correspondence between observed data and solution values. One symbol is drawn for each observation point at the intersection of the observed and computed values for the point. This plot can show the trend of the solution values with regard to matching the observed data. Only those points whose value is specified as observed for the selected data type will be shown in the plot. These plots are created in the Plot Wizard by setting the Plot Type to Computed vs. Observed. A sample plot is shown in the figure.



Computed vs. Observed Plot Options

The second step contains the following options.

Coverage and Measurements

The observation coverage and measurements can be selected for each plot. If only one observation coverage and measurement exist these are used by default.

Use current solution

This option causes the plot to compare the observed values with the values of the current solution and time step for each observation point. When the active solution changes, the plot is recomputed and updated. If only one solution is in memory then this option is defaulted.

Use selected solutions

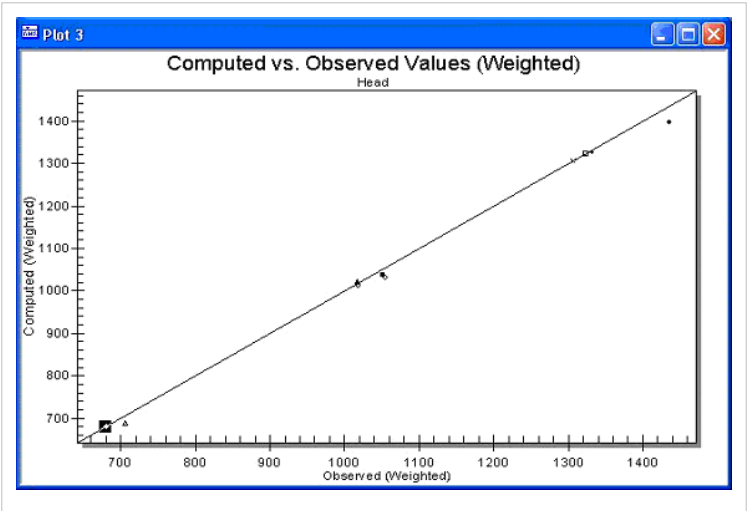
This option causes the plot to compare the observed values with the values of the specified solution for each observation point. Changing the active solution does not affect the plot.

Use time step

Because the plot shows only values from a specific time step, The Active Times Step is used by default.

Computed vs. Observed (Weighted)

A Computed vs. Observed Weighted plot is used to display how well the entire set of weighted observed values match a model solution. These weights are set by selecting the Observation item in the MODFLOW menu. A 45^{o} line is drawn on this plot, which represents a perfect correspondence between observed data and solution values. One symbol is drawn for each observation point at the intersection of the weighted observed and computed values for the point. This plot can show the trend of the solution values with regard to matching the weighted observed data. Only those points whose value is specified as observed for the selected data type will be shown in the plot. This plot is not available with transient data. These plots are created in the Plot Wizard by setting the Plot Type to Computed vs. Observed (Weighted). A sample plot is shown in the figure.



Computed vs. Observed Weighted Plot Options

After the plot type is set in the First Page of the Plot Wizard, the Next button is clicked to open the Computed vs. Observed Weighted Plot Options wizard page. This page contains the following:

Coverage and Measurements

The observation coverage and measurements can be selected for each plot. If only one observation coverage and measurement exist these are used by default.

Use current solution

This option causes the plot to compare the weighted observed values with the values of the current solution and time step for each observation point. When the active solution changes, the plot is recomputed and updated. If only one solution is in memory then this option is defaulted.

Use selected solutions

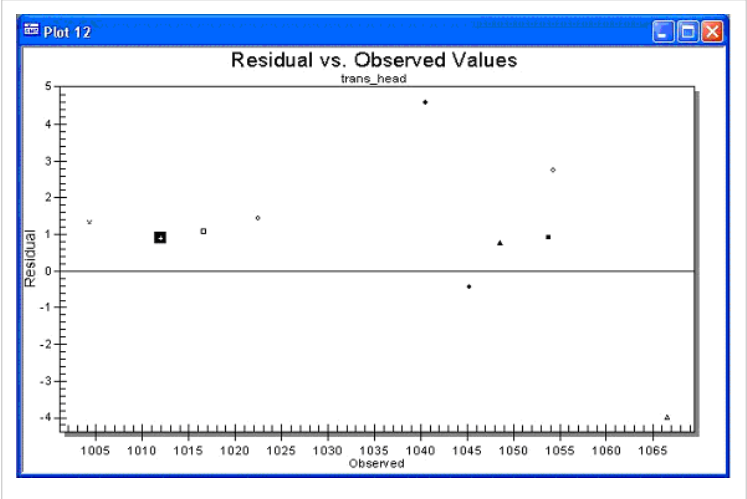
This option causes the plot to compare the weighted observed values with the values of the specified solution for each observation point. Changing the active solution does not affect the plot.

Use time step

Because the plot shows only values from a specific time step, The Active Times Step is used by default.

Residual vs. Observed

A Residual vs. Observed plot is used to display how well the entire set of observed values match a model solution. On this plot is drawn a horizontal line along an error of zero, representing what would be a perfect correspondence between observed data and solution values. Then, one symbol is drawn for each observation point at the intersection of the observed and residual (computed-observed) values for the point. This plot can show the trend of the solution values with regards to matching the observed data. Only those points whose value is specified as observed for the selected data type will be shown in the plot. These plots are created in the Plot Wizard by setting the Plot Type to Residual vs. Observed. A sample plot is shown in the figure.



Residual vs. Observed Plot Options

After the plot type is set in the First Page of the Plot Wizard, the **Next** button is clicked to open the Residual vs. Observed Plot Options wizard page. This page contains the following:

Coverage and Measurements

The observation coverage and measurements can be selected for each plot. If only one observation coverage and measurement exist these are used by default.

Use current solution

This option causes the plot to compare the observed values with the residual values of the current solution and time step for each observation point. When the active solution changes, the plot is recomputed and updated. If only one solution is in memory then this option is defaulted.

Use selected solutions

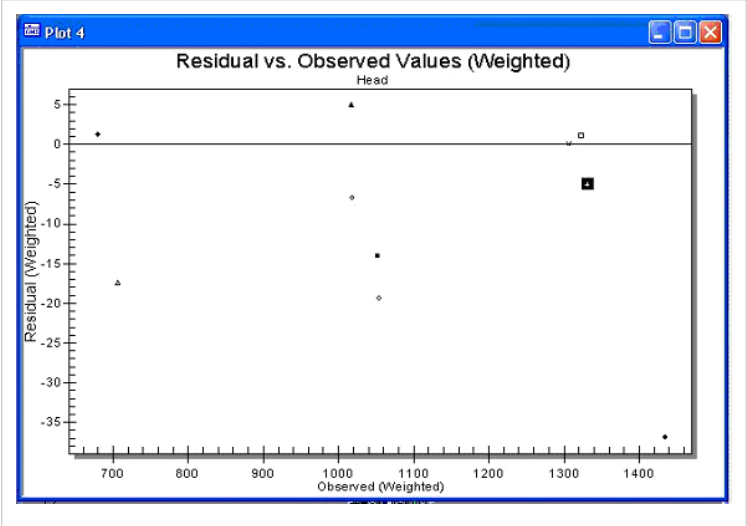
This option causes the plot to compare the observed values with the residual values of the specified solution for each observation point. Changing the active solution does not affect the plot.

Use time step

Because the plot shows only values from a specific time step, The Active Times Step is used by default.

Residual vs. Observed (Weighted)

A Residual vs. Observed (Weighted) plot is used to display how well the entire set of weighted observed values match a model solution. On this plot is drawn a horizontal line along an error of zero, representing what would be a perfect correspondence between weighted observed data and solution values. One symbol is drawn for each observation point at the intersection of the weighted observed and residual (computed-observed) values for the point. This plot can show the trend of the solution values with regards to matching the weighted observed data. Only those points whose value is specified as observed for the selected data type will be shown in the plot. This plot is not available with transient data. These plots are created in the Plot Wizard by setting the Plot Type to Residual vs. Observed (Weighted). A sample plot is shown in the figure.



Residual vs. Observed Plot Options

After the plot type is set in the First Page of the Plot Wizard, the Next button is clicked to open the Residual vs. Observed (Weighted) Plot Options wizard page. This page contains the following:

Coverage and Measurements

The observation coverage and measurements can be selected for each plot. If only one observation coverage and measurement exist these are used by default.

Use current solution

This option causes the plot to compare the weighted observed values with the residual values of the current solution and time step for each observation point. When the active solution changes, the plot is recomputed and updated. If only one solution is in memory then this option is defaulted.

Use selected solutions

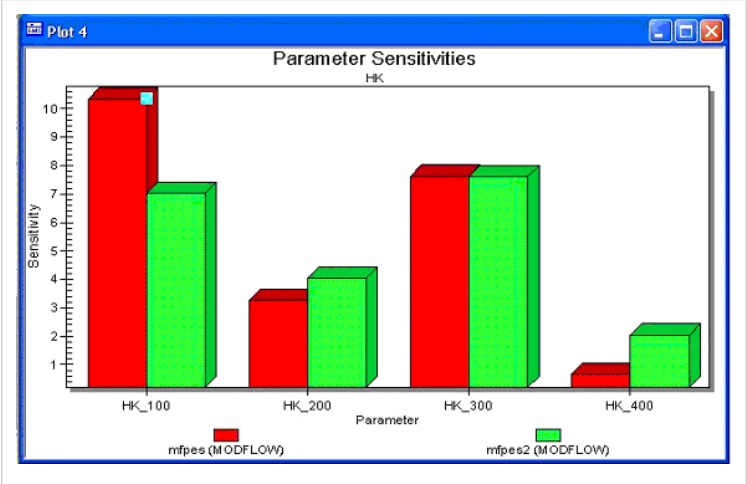
This option causes the plot to compare the weighted observed values with the residual values of the specified solution for each observation point. Changing the active solution does not affect the plot.

Use time step

Because the plot shows only values from a specific time step, The Active Times Step is used by default.

Parameter Sensitivity

A Parameter Sensitivity plot is used to display the sensitivity of the MODFLOW parameters. These plots are created in the Plot Wizard by setting the Plot Type to Paramater Sensitivity. A sample plot is shown in the figure.



Parameter Sensitivity Plot Options

After the plot type is set in the First Page of the Plot Wizard, the **Next** button is clicked to open the Parameter Sensitivity Plot Options wizard page. This page contains the following:

Parameter Type

This option changes the parameter type displayed.

Use current solution

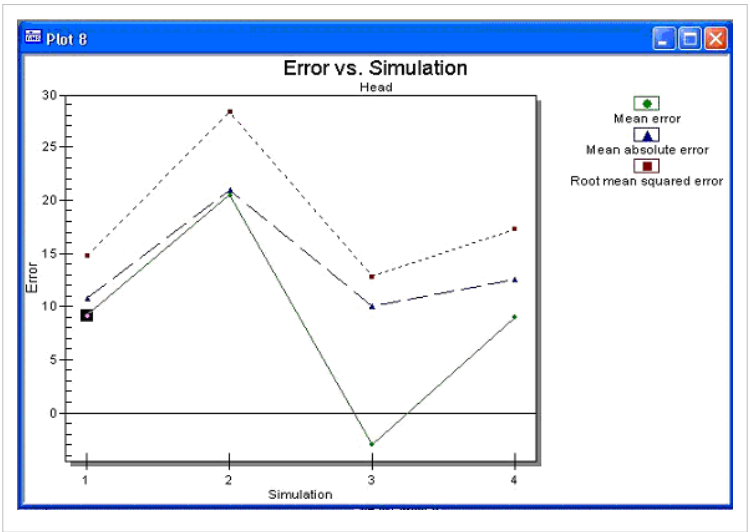
This option causes the plot to compare Parameter Sensitivities of the current solution. When the active solution changes, the plot is recomputed and updated. If only one solution is in memory then this option is defaulted.

Use selected solutions

This option causes the plot to compare Parameter Sensitivities of the specified solution. Changing the active solution does not affect the plot.

Error vs. Simulation

An Error vs. Simulation plot is generally used with steady-state simulations and measurement types. It may be used in transient simulations. This plot can display the mean error, mean absolute error, and root mean squared error between successive solutions and observed data. Various simulations would be run after changing model parameters, such as hydraulic conductivity or recharge. The plot will show trends in the solution to see if model parameter changes are causing better calibration with measured field data. Error vs. Simulation plots are created in the Plot Wizard by setting the Plot Type to Error vs. Simulation. A sample plot is shown in the figure.



Error vs. Simulation Plot Options

After the plot type is set in the First Page of the Plot Wizard, the **Next** button is clicked to open the Error vs. Simulation Plot Options wizard page. This page contains the following:

Solutions

This lists all available solutions.

Move Up/Move Down

GMS initially shows the solutions in the order they were opened. However, this is not necessarily the order in which they were run. To change the order, highlight a solution and move it up or down to rearrange their order.

Observed

This lists all available measurement types that were set up in the Observation Coverage Options dialog, both constant and transient. This plot is generally performed using constant solutions and measurement types, although transient data is allowed. You should select in the combo box the measurement type that you are comparing.

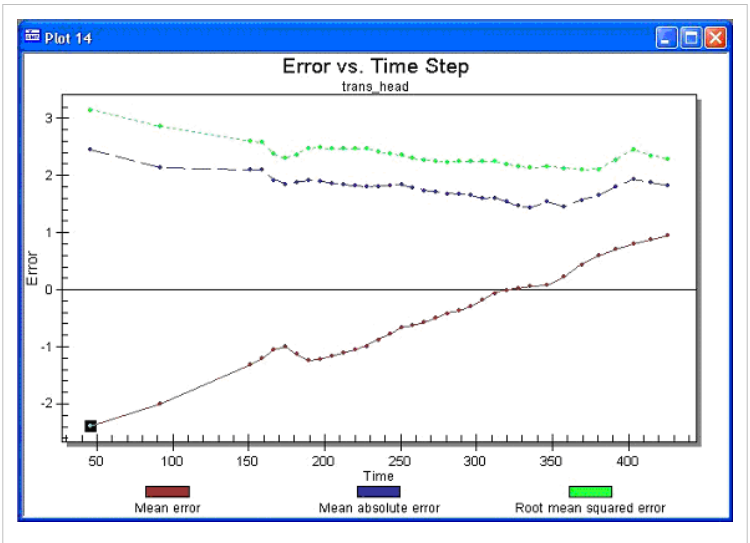
Check Box Options

There are three options that can be turned on or off. They determine whether the mean error, mean absolute error, and root mean squared error plots should be shown. Because these values are an average of all observation points, their line and symbol styles are not linked to any one observation point, but can be defined by clicking on the appropriate canvas window in the dialog.

Error vs. Time Step

An Error vs. Time Step plot is used with transient simulations to display the mean error, mean absolute error, and root mean squared error between a solution and observed data as a function of time. This plot applies to a single Data Set in a model solution. Transient measurement types will show the average errors at each time step of the data set. Error vs. Time Step plots are created in the Plot Wizard by setting the Plot Type to Error vs. Time Step. A sample plot is shown in the figure.

Beginning with version 8.0, this plot can no longer be used with MODFLOW.



Error vs. Time Step Plot Options

After the plot type is set in the First Page of the Plot Wizard, the **Next** button is clicked to open the Error vs. Time Step Plot Options wizard page. This page contains the following:

Use current solution

This option causes the plot to compare Parameter Sensitivities of the current solution. When the active solution changes, the plot is recomputed and updated. If only one solution is in memory then this option is defaulted.

Use selected solutions

This option causes the plot to compare Parameter Sensitivities of the specified solution. Changing the active solution does not affect the plot.

Observed

This lists all available measurement types that were set up in the *Observation Coverage Options* dialog, both constant and transient. A constant measurement should be used for solutions with only one time step, while a transient measurement should be used for transient solutions. You should select the measurement type in the combo box that you are comparing. If only one exists then it is used by default.

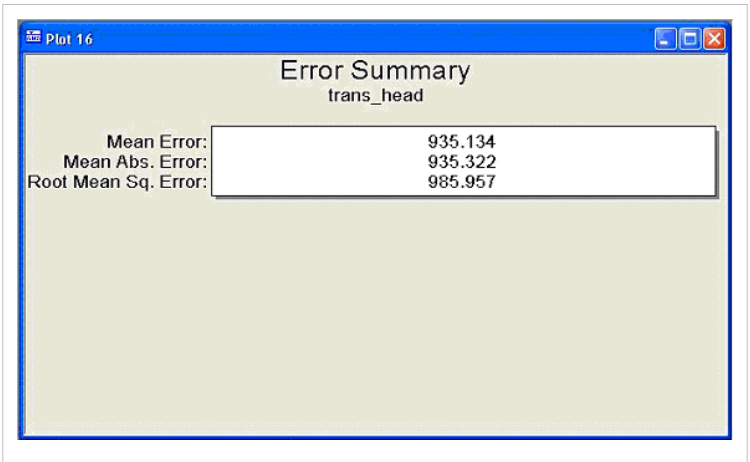
Check Box Options

There are three options that can be turned on or off. They determine whether the mean error, mean absolute error, and root mean squared error plots should be shown. Because these values are an average of all observation points, their line and symbol styles are not linked to any one observation point, but can be defined by clicking on the appropriate canvas window in the dialog.

Error Summary

See the Error Summary Plot page for more details. An Error Summary plot is used to display the mean error, mean absolute error, and root mean squared error for a Solution. Error Summary plots are created in the Plot Wizard by setting the Plot Type to Error Summary. A sample plot is shown in the figure.

Beginning with version 8.0, this plot can no longer be used with MODFLOW. This information can be found by right-clicking on the MODFLOW solution in the Project Explorer.



Error Summary Plot Options

After the plot type is set in the First Page of the Plot Wizard, the **Next** button is clicked to open the Error Summary Plot Options wizard page. This page contains the following:

Use active solution

This option causes the plot to show a summary of mean error between the set of observed values with the values of the active solution for each observation point. When the active solution changes, the plot is recomputed and updated. This option is always on by default at this time

All time steps

This option shows the mean error for all time steps instead of only at the current or specified time step value.

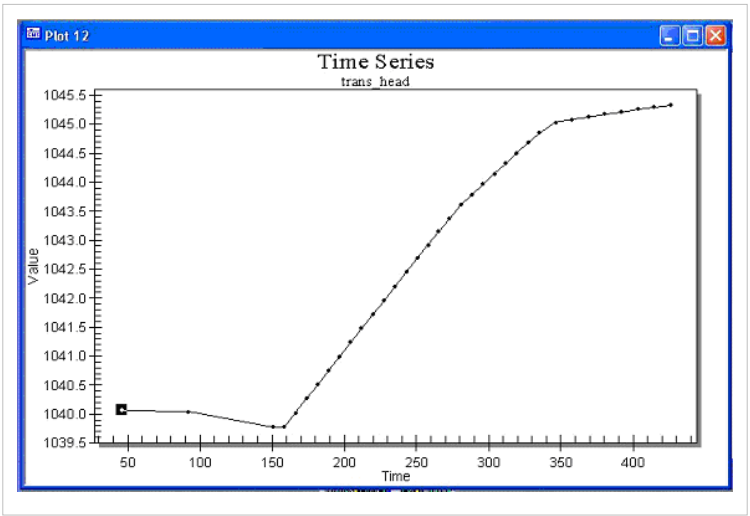
Observed

This section shows the available list of observed values that were set up in the *Observation Coverage Options* dialog. Both constant and transient observed values should be available in this dialog.

Time Series

A Time Series plot is used to display the time variation of one or more scalar Data Sets associated to a given point inside a model solution. In addition, if transient calibration data has been defined, a band can be shown which represents a time variant Calibration Target. Only transient data sets may be used in these plots. Time Series plots are created in the Plot Wizard by setting the Plot Type to Time Series. A sample plot is shown in the figure.

Beginning with version 8.0, these plots have been changed to show observation targets instead of a "band" around the observed data.



Time Series Plot Options

After the plot type is set in the First Page of the Plot Wizard, the Next button is clicked to open the Time Series Plot Options wizard page. This page contains the following:

Coverage and Measurements

The observation coverage and measurements can be selected for each plot. If only one observation coverage and measurement exist these are used by default. All Measurements can also be selected to plot all of the different measurements on one plot.

Use current solution

This option causes the plot to display the values of the active Solution for each observation point being plotted. When the active solution changes, the plot is recomputed and updated. If only one solution is in memory then this option is defaulted.

Use selected solutions

This option causes the plot to display the values of one or more specified Solutions for each point being plotted. Changing the active solution does not affect the plot.

Selected Points

Each of the observation points are shown in a spreadsheet. The show toggle determines which Observation Points will be displayed in a Time Series Plot in the Plot Window. This way, only certain points are plotted in each individual time series plot. All the points can either be turned on or off by selecting the corresponding buttons below the spreadsheet.

Calibration Target

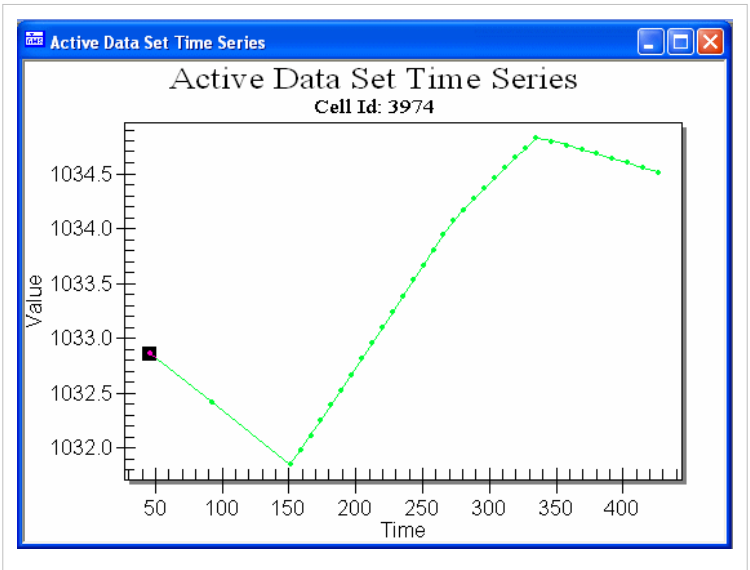
Turn this on to display a band in the plot to define the valid calibration range. If this is turned off, only the calibration curve will be drawn.

Time interval

A starting time and an ending time of observations can be specified by two combo boxes. Only the interval chosen will be shown on the time series plot.

Active Dataset Time Series

A Time Series plot is used to display the time variation of one or more scalar Datasets associated to a given point selected in a model solution. Time Series plots are created in the Plot Wizard by setting the Plot Type to Active Dataset Time Series. A sample plot is shown in the figure.



Active Dataset Time Series Plot Options

After the plot type is set in the First Page of the Plot Wizard, the **Finish** button is clicked to create the active time series plot. For 2D and 3D grids select a grid cell and for 2D and 3D meshes select a node to display the scalar values of the dataset at that location over time. You can select up to five locations to be plotted on one plot.

S/S Flow vs. Time

A Flow vs. Time plot is used to display the flow or water over time for a selected feature object or grid cell. Flow vs. Time plots are created in the Plot Wizard by setting the Plot Type to Flow vs. Time. A sample plot is shown in the figure.

S/S Flow vs. Time Plot Options

After the plot type is set in the First Page of the Plot Wizard, the **Finish** button is clicked to create the flow vs time plot. This plot will report the flow either by selecting a feature object defined as a MODFLOW source/sink in the map module or by selecting a group of cells in the 3D Grid. To change the plot options for the Flow vs Time Plot right-click on the plot and select the ***Plot Data*** command. The *Plot Data* dialog contains the following:

Feature Objects flow

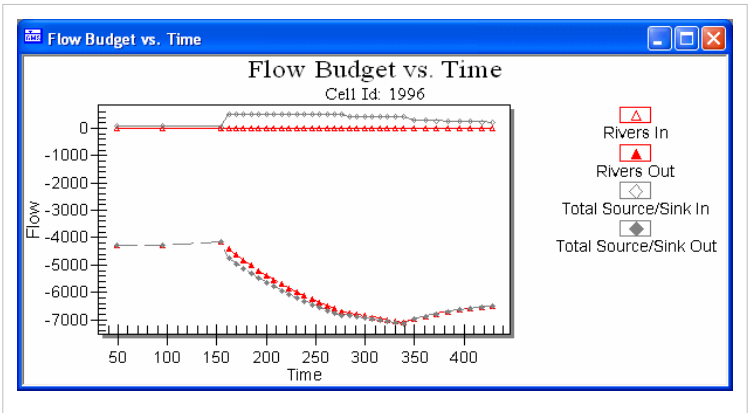
This option allows you to select what data to plot when a Source/Sink feature object is selected in the Map module. By default the computed flow is plotted. If observed values exist for the feature object than those values can also be plotted along with calibration target ranges.

Grid Cell Flow

This option allows you to select what Source/Sink flows to be plotted when Grid Cells are selected in the 3D grid module. The options are: Storage, Constant Head, Drains, General Heads, Rivers, Streams, Recharge, Evapotranspiration, and Total Source/Sink flow in and out of the cells.

Flow Budget vs. Time

A Flow Budget vs. Time plot is used to display the flow or water over time for selected grid cell or for zone budget ids. Flow Budget vs. Time plots are created in the Plot Wizard by setting the Plot Type to Flow Budget vs. Time. A sample plot is shown in the figure.



Flow Budget vs. Time Plot Options

After the plot type is set in the First Page of the Plot Wizard, the **Next** button is clicked to open the Flow Budget Plot Options wizard page. The options on this page are discussed below. The **Finish** button is clicked to create the flow budget vs time plot. This plot will report the flow either by selecting a group of cells in the 3D Grid or by using the zone budget ids.

Display using

This option allows you to determined the data to plot. The Flow budget will be computed either from the selected 3D grid cells or from the selected zone budget ids in the spreadsheet.

Grid Cell Flow

This option allows you to select what Source/Sink flows to be plotted. The options are: Storage, Constant Head, Drains, General Heads, Rivers, Streams, Recharge, Evapotranspiration, and Total Source/Sink flow in and out of the cells.

Gage Package Value vs. Time

This plot shows a time series for a single gage package column. After the plot type is set in the First Page of the Plot Wizard, the Next button is clicked to open the Gage Package Time Series wizard page. To generate a gage plot the gage file needs to be selected from the list on the left, and the data column from the list on the right. If GMS is unable to properly read the data from the gage file, an error is shown in the wizard page to the right of the data column list.

Finish

When the Finish button is selected, a window is opened with the plot. Plot windows are created each time the Plot Wizard is run. See the 2D Plots discussion for information on editing plots that have been created.

Right-Click Menu Commands

Once the plot has been created, the appearance and data of the plot can be altered with commands accessed by right-clicking on the plot window. The commands available through the right-click menu are described below.

Plot Data – The plot data command brings up the second step of the plot wizard where the attributes of each plot are set. The plot data can be updated based upon the type of plot. For Histogram plots it brings up a dialog used to edit the precision displayed on the plot.

Display Options – The Display Options command allows you to change general display attributes of the plot (Border Style, Precision, Axis type, Fonts, Colors, etc.).

Axis Titles – The axis titles menu is used to change the x and y axis titles.

Set As Display Defaults – To save the plots current settings as the default settings, select the Save Settings menu item.

Legends – The legend can be turned on by right clicking on the plot and selecting a location from the Legend submenu. The maximum number of points included in the legend depends upon the size of the plot window.

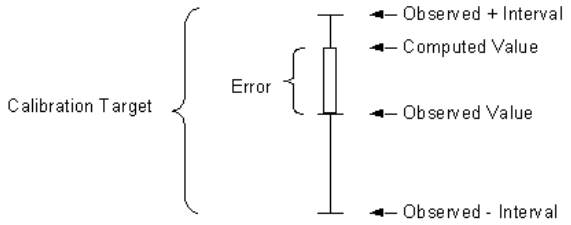
Export/Print – By selecting on the Export/Print menu item, a dialog is displayed showing options for printing or exporting the plots values.

Default Display – To save the plots current settings as the default settings, select the Save Settings menu item.

View Values – To view the values of the plot in a spreadsheet format, select the View Values menu. A spreadsheet containing the data points associated with the current plot appear.

Calibration Targets

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS



Example of a calibration target

If an observed value has been assigned to an observation point or if an observed flow has been assigned to an arc or polygon, the calibration error at each object can be plotted using a "calibration target". A set of calibration targets provides useful feedback on the magnitude, direction (high, low), and spatial distribution of the calibration error.

The components of a calibration target are illustrated in the following figure. The center of the target corresponds to the observed value. The top of the target corresponds to the observed value plus the interval and the bottom corresponds to the observed value minus the interval. The colored bar represents the error. If the bar lies entirely within the target, the color bar is drawn in green. If the bar is outside the target, but the error is less than 200%, the bar is drawn in yellow. If the error is greater than 200%, the bar is drawn in red. The display options related to calibration targets are specified in the *Feature Object Display Options* dialog.

If the active time step is before the first observed time, or after the last observed time, the targets are drawn lighter.

Model Checker

Once a mesh or grid is generated and all of the analysis options and boundary conditions have been specified, the next step is to save the simulation to disk and run the model. However, before saving the simulation and running the model, the model should be checked with the Model Checker. Because of the significant amount of data required for a simulation for all the different models, it is often easy to neglect important data or to define inconsistent or incompatible options and parameters. Such errors will either cause the model to crash or to generate an erroneous solution. The purpose of the Model Checker is to analyze the input data currently defined for a model simulation and report any obvious errors or potential problems. Running the Model Checker successfully does not guarantee that a solution will be correct. It simply serves as an initial check on the input data and can save a considerable amount of time that would otherwise be lost tracking down input errors.

Running the Model Checker

To check the current data, select the ***Check Simulation*** command from the "current model" menu. The *Model Checker* dialog will appear. To run the Model Checker, select the button labeled **Run Check** at the top of the dialog. This generates a list of possible errors and warning messages in the top scrolling window.

If the model checker finds any potential errors with your input, then fix the errors and rerun the model checker.

Options

The Checker Options button in the Model Checker allows you to customize the checks that will be performed. A check box is provided for each category of the model input data. Turning off any of the options will suppress the warnings and errors associated with these categories. Also provided is the option to define the maximum number of errors and warnings to be reported of the same type. Messages classified as warnings can also be suppressed.

Save Messages

The Save Messages button provides the option to save the current listing of warnings and error messages to a text file.

One unique feature of the MODFLOW Model Checker is the Fix Layer Errors option.

7.2. Parameters

Parameters

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

A parameter is a variable that is used to control one or more other values. GMS uses parameters with MODFLOW to define inputs for forward runs or for model calibration or for stochastic modeling.

Key value and standard MODFLOW parameters

There are two ways to do MODFLOW parameterization in GMS:

- The key value method (recommended)
- The standard MODFLOW method

Key values

Key values are used to parameterize the MODFLOW input. Key values can be used for both the zonation and pilot points methods of defining parameters. A key value is a number that is assigned as input that marks the input values as belonging to a parameter. When selecting a key value, a number should be chosen that is not likely to occur in typical input for that parameter. We recommend using negative numbers in most cases.

For example, a model may have four K zones and three recharge zones. The four K zones would be marked by assigning -100, -200, -300, and -400 to the cells in the K array for each of the four zones. Likewise, the recharge zones would be marked by assigning -500, -600, and -700 to the appropriate cells in the recharge array.

Key values can be assigned either by editing the cell array values directly in the 3D grid module or by assigning the values to a set of polygons or arcs in the conceptual model and selecting the ***Map → MODFLOW*** command.

When MODFLOW is running and data is read from the HDF5 file, if one of the parameter key values is found in the data then the parameter value is substituted into the array or list.

The key value approach makes it much easier to support transient parameters and very large sets of pilot points. Also, this approach allows GMS users to use parameters with the BCF package. This feature is not available with the MODFLOW PES process.

Standard MODFLOW parameters

The "key value" approach is the preferred approach to defining parameters in GMS. Using key values is the only option for WEL, RIV, DRT, DRN, GHB, CHD, STR, and HFB parameters. In GMS, using the key value approach with these parameters provides the same functionality of parameter instances available in MODFLOW.

Key values can also be used with array based parameters. However, GMS does support defining array based parameters with clusters and instances (for ETS, EVT, RCH parameters).

When reading in a MODFLOW simulation that was created outside of GMS all parameters will be converted to key values so long as the parameter can be represented by key values.

Key values can not be used to represent an array based parameter if the parameter is defined using more than one cluster or more than one instance. Also, if more than one parameter uses the same zone array with a matching IZ value then the parameter can not be represented using key values. Further, if the multiplier array associated with the parameter has values other than 1.0 in the zones where the parameter is used then the parameter can not be represented with key values.

The standard MODFLOW method is considered an "advanced" feature in GMS. **Only users that understand how MODFLOW uses parameters should attempt to use this feature.**

For more information, refer to the Standard MODFLOW Parameters page.

Using parameters

To use parameters, you must:

1. Parameterize your model and
2. Create a parameter list

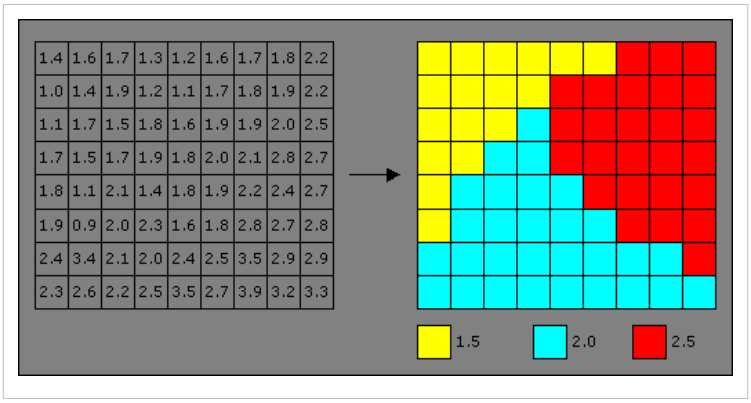
Parameterizing the model

There are two methods used to parameterize a model:

- Zonation
- Pilot Points

Zonation

The most common form of parameterization is zonation. With this approach, an array of input values is divided into zones where all of the cells in each zone share a single parameter value. This concept is illustrated in the figure below. The entire array of values is represented by three parameters.



Zonation is most easily accomplished in GMS by assigning key values to polygons in the map module. It can also be accomplished by assigning key values to a selected set of cells in the 3D grid module using the ***Cell Attributes*** command.

Pilot points

Another way to parameterize a model is to use pilot points. For more information on pilot points, refer to the Pilot Points page.

Creating a parameter list

The list of parameters is defined using the *Parameter* Dialog. In general, the number of parameters should be less than the number of observations. However, if the user chooses to use pilot points with PEST in regularization mode, then the number of parameters does not have to be less than the number of observations.

Legal MODFLOW Parameters

The following MODFLOW input values can be defined as parameters in GMS:

Package	Option
LPF/HUF	• HK – Horiz. hydraulic conductivity • HANI – Horiz. anisotropy • VK – Vert. hydraulic conductivity • VKCB – Vert. hydraulic conductivity of confining beds • VANI – Vert. anisotropy • SS – Specific storage • SY – Specific yield
Recharge	RCH – Recharge value
Evapotranspiration	EVT – Max ET flux
Evapotranspiration Segments	ETS – Max ET flux
Well	WELL – Pumping rate
River	RIV – River conductance
General Head	GHB – Conductance
Drain	DRN – Conductance
Time-Variant Specified-Head	CHD – Head

Stream	STR – Conductance
Stream-flow Routing	SFR – Conductance
Horizontal Flow Barrier	HFB – Hydraulic characteristic

Parameter Dialog

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR
Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

When building a MODFLOW inverse model, the input data must be parameterized. This is accomplished by assigning a set of key values to selected input fields. Once the key values are assigned, the next step is to create a parameter list.

Parameters can also be used to perform forward runs.

See also Standard MODFLOW Parameters.

Creating/Deleting Parameters

The *Parameters* dialog contains the list of parameters. A new parameter can be created by selecting the **New Parameter** button. Each parameter that is defined should correspond to a key value that has been defined in the MODFLOW input. A parameter can be removed from the list by selecting the parameter and selecting the Delete Parameter button. The entire list of parameters can be deleted by selecting the Delete All button.

Initialize from Model

In most cases, the fastest and simplest way to create the parameter list is to use the **Initialize from Model** button. When this button is selected, GMS traverses the MODFLOW input data corresponding to legal parameter values and searches for key values. It is assumed that the key values are entered as negative numbers. When a unique negative number is found, a new parameter is added to the list, and a default name is given to the parameter based on the parameter type.

NOTE: The ***Initialize from Model*** command will not search for key values in the Well package. This is because negative pumping rates are perfectly common and do not necessarily correspond to key values. If you wish to define a well Q as a parameter, you should use the **New Parameter** button and manually create the parameter.

Import Optimal Values

After performing an inverse model run, the inverse code writes out a text file containing the set of optimal parameter values corresponding to the minimum calibration error. The next step is to read these values into GMS. This is accomplished by selecting the **Import Optimal Values** button. Once you open the file, the optimal parameter values will be loaded into the starting value field and displayed in the parameter list.

If any of your parameters use pilot points, importing the optimal values will also import and create a new data set for the associated 2D scatter point set.

Spreadsheet

The spreadsheet holds parameters, their types, starting and bounding values and other options that depend on the packages and run options defined for the current MODFLOW simulation in the Global Options\Basic Package.

Parameter Estimation Solve

Toggle on this option to solve for the optimal parameter value using a parameter estimation program like PEST. (See Automated Parameter Estimation)

Name

The name must be unique and limited to 8 characters. The default name should be sufficient. This name will be used to reference the parameter in the MODFLOW solution and output files.

Key

The key value, usually a negative value, must be unique and is used to link the parameter with the data in the MODFLOW input files. The key value should also be entered into the MODFLOW data by directly entering the values using the grid based approach or by using the map module.

Type

GMS supports most of the parameter types that are also supported by the MODFLOW PES process.

Start Value/Pilot Points

This value will be the starting value for inverse modeling, the mean value for stochastic modeling, or the substitute value for forward runs.

For inverse modeling, the closer the starting value is to the "optimal" value, the better the odds that the inverse model will converge and the less time it will take to converge. It is generally not a good idea to give all parameters of a given type (e.g., recharge) a constant value and let the inverse model start from that point. Ideally, field tests, soil types, ground cover, and sound modeling judgement can provide a good set of starting values. It is also a good idea to undergo some manual trial-and-error calibration prior to setting up the inverse code.

Pilot points can be used to define a parameter by selecting the drop-down arrow in this column and selecting the **Pilot points** option. Pilot points are an alternative to using zonation to define parameter locations. When you use pilot points, you use the Pilot Point Options button to choose a 2D point scatter point set and choose the appropriate interpolation options.

Min Value Max Value

The min and max values will provide the bounds for the parameter, and they must encompass the starting value. The parameter values will be forced between these values during inverse and stochastic modeling. In these cases, the min and max values are just "suggestions" and are used to predict parameter values.

When selecting the min and max values for the parameters, care should be taken not to make the range in values too large. Inverse models are highly senstive to the stability of the underlying model. If MODFLOW does not converge, the invese model will not be able to find a solution. Furthermore, excessive cell drying can cause the inverse model to fail to converge. If the min and/or max values are too extreme, the odds of the MODFLOW model not converging or excessive numbers of cells going dry increases. It is best to select a limited range for each parameter and then to compare the final optimal parameter value to this range. If the optimal value is at either the min or max of the range, the range can be adjusted and the inverse model can be re-launched.

Log xform

This option log transforms the value during prediction process of inverse modeling and the random number generation process of stochastic modeling. The best parameters to log transform are those that can vary by orders of magnitude like hydraulic conductivity.

It is also recommended that you log transform recharge parameters if you are using pilot points for hydraulic conductivity and the hydraulic conductivity parameter is log transformed.

BSCAL

MODFLOW documentation includes:

This value is an alternate scaling factor for the parameter, and always needs to be a positive number. If the parameter value becomes 0.0, which can occur for parameters that are not log transformed, BSCAL is used in the scaling. If the absolute value of the parameter is less than BSCAL, BSCAL is used in the scaling. The best value to use is problem dependant. Good choices are the smallest reasonable value of the parameter or a value two to three orders of magnitude smaller than the value specified by the starting value. If the smallest reasonable value is 0.0, a reasomable non-zero value needs to be used. BSCAL has no effect on the scaled sensitivities for log-transformed parameters.

Multiplier

Select this option to include a multiplier array for RCH and HK parameters.

Data Set / Folder

Use this button to select a multiplier array of a set of multiplier arrays (stochastic only) by selecting a folder of data sets.

Stochastic Options

Standard Deviation

Use this field to specify the standard deviation of a parameter for a stochastic simulation.

Mean value

Use this field to specify the mean of a parameter for a stochastic simulation.

Distribution

When the parameter is stochastic, use this option to choose between a normal or linear distribution. A random number for the parameter is generated using the distribution, the mean value (starting value) and the standard devation.

Std Deviation

The standard deviation is used for a stochastic parameter to generate a random number using the chosen distribution.

Num Segments

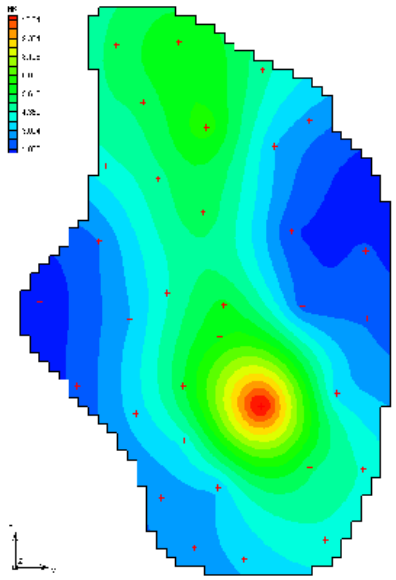
When the parameter is stochastic and the stochastic method is Latin Hypercube, the number of segments helps determine how man total MODFLOW runs will be used.

Pilot Points

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

Pilot points can be thought of as a 2D scatter point set. Instead of creating a zone and having the inverse model estimate one value for the entire zone, the value of the parameter within the zone is interpolated from the pilot points. Then the inverse model estimates the values at the pilot points. The figure below shows a set of pilot points used to estimate horizontal hydraulic conductivity. Notice how the hydraulic conductivity now varies from cell to cell. When the inverse model runs, the values at the pilot points are adjusted and the “surface” defining the variation of K values is warped until the objective function is minimized.

PEST provides an additional option for the pilot point method called “regularization”. Regularization imposes an additional measure of “stiffness” to the parameter being interpolated via a “homogeneity” constraint. In the absence of any strong influence from the PEST objective function, this constraint causes values at pilot points to approximate the mean value of adjacent pilot points. This constraint makes the inversion process much more stable and makes it possible to violate one of the typical constraints associated with parameter estimation: namely, the requirement that the number of parameters must be less than the number of observations. With regularization, the number of parameters can greatly exceed the number of observations. As a result, complex hydraulic conductivity distributions can be defined, resulting in extremely low residual error. The pilot point method with regularization is an incredibly powerful feature of PEST. (For more on PEST see Automated Parameter Estimation)



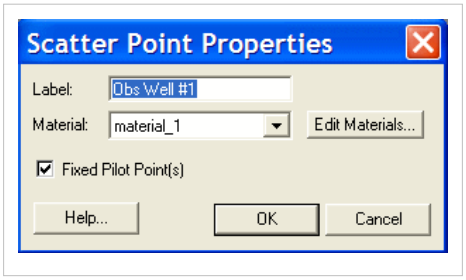
Pilot Points and Resulting Conductivity Field

Interpolation Options

Kriging and IDW are the only interpolation options supported with pilot points. The kriging option requires the establishment of a model variogram (by creating a nested structure). The nodal function options in the IDW method are not supported because those schemes compute gradients based on the data set values at the surrounding points. With the pilot point method, the values at the points will change during the inversion process, thereby rendering the previously computed gradients inaccurate.

Pilot Point Conditioning

For pilot point interpolation of hydraulic conductivity, it is sometimes useful to include one or more measured K values with the pilot point set. These measured values could represent K values extracted from a field pump test. The Pilot Point Conditioning option is available to represent this scenario. One of the properties associated with scatter points is a **Fixed pilot point** toggle. If this toggle is on and the corresponding scatter point set is used for pilot point interpolation, the K value assigned to the point is not allowed to vary during the parameter estimation process.



Guidelines for Placement of Pilot Points

Note: The recommended maximum number of pilot points is about 200.

1. Place points between observations rather than on top of observations
2. Add greater density where there are more observations
3. Add points where head gradient is steep
4. Place a row of points between observation wells and head-dependent boundaries
5. Fill in the gaps

Multiplier Arrays for Parameters

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

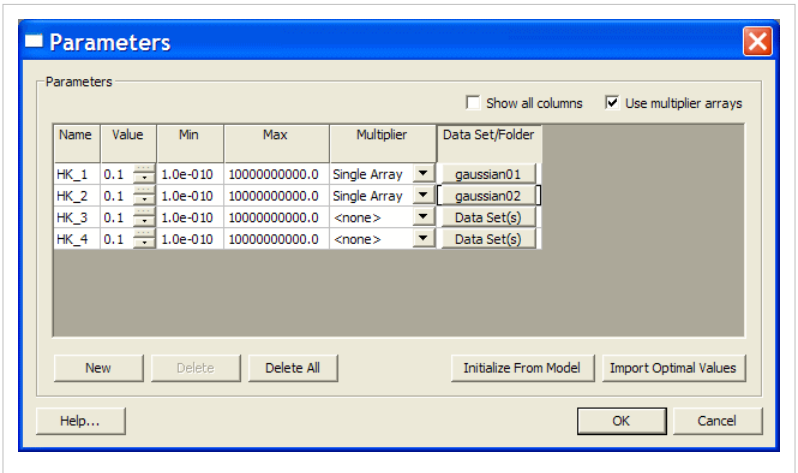
HK and RCH parameters may be associated with a multiplier array. A data set is associated with a parameter zone and the starting parameter value is multiplied by the data set to define the spatial variation of the parameter throughout the zone.

Setting up Multiplier Arrays

The multiplier data set is associated with the parameter zone in the Parameters dialog by first turning on the **Use multiplier arrays** toggle, selecting either **Single Array** or **Multiple Array** from the pull-down list in the **Multiplier** column, and then selecting the data set using the button in the **Data Set/Folder** column. If the **Single Array** option is selected, a single data set should be selected in the Data Set/Folder column. If the **Multiple Array** option is selected, an entire folder of data sets should be selected. The **Multiple Array** option is only available for stochastic modeling.

If the **Single Array** option is selected the array for the parameter will be defined as the starting/mean value multiplied by the multiplication array for all cells associated with the parameter. Multiplier arrays are not available for parameters that use pilot points.

When the **Multiple Arrays** Option is selected, the number of stochastic model runs will be a function of how many data sets are in the folder selected by the user for the multipliers.



Guassian Field Based Monte Carlo Simulations

The **Multiple Array** option for user-defined multiplier arrays can be used in combination with the new Gaussian Field Generator to perform a Monte Carlo simulation. This is accomplished by first generating a set of Gaussian fields using the FIELDGEN code and then associating the resulting folder of data sets with a parameter as described above. A stochastic simulation can then be performed using the Parameter Randomization method. For each model instance, GMS loads a new instance of the multiplier array into the model input. A default parameter value of 1.0 is generally recommended in such instances.

Standard MODFLOW Parameters

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR

Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

Caution

There are two ways to do MODFLOW parameterization in GMS:

1. The key value method **(recommended)**
2. The standard MODFLOW method

The "key value" approach is the preferred approach to defining parameters in GMS. Using key values is the only option for WEL, RIV, DRT, DRN, GHB, CHD, STR, and HFB parameters. In GMS, using the key value approach with these parameters provides the same functionality of parameter instances available in MODFLOW.

Key values can also be used with array based parameters. However, GMS does support defining array based parameters with clusters and instances (for ETS, EVT, RCH parameters).

When reading in a MODFLOW simulation that was created outside of GMS all parameters will be converted to key values so long as the parameter can be represented by key values.

Key values can not be used to represent an array based parameter if the parameter is defined using more than one cluster or more than one instance. Also, if more than one parameter uses the same zone array with a matching IZ value then the parameter can not be represented using key values. Further, if the multiplier array associated with the parameter has values other than 1.0 in the zones where the parameter is used then the parameter can not be represented with key values.

The standard MODFLOW method is considered an "advanced" feature in GMS. **Only users that understand how MODFLOW uses parameters should attempt to use this feature.**

This article describes how to use the standard MODFLOW parameterization method in GMS.

Support in GMS

Reading

GMS can read MODFLOW models that use standard MODFLOW parameters.

Writing

GMS writes MODFLOW models using the key value approach. So, although GMS can read models that use standard MODFLOW parameters, it cannot write them using the standard MODFLOW parameter method. Writing is usually done using the key value method. There are a few exceptions which include recharge and evapotranspiration parameter instances, and LPF and HUF parameter clusters. GMS will write these parameters using the standard MODFLOW parameter method because these cannot always be represented using the key value method.

Editing

When GMS reads a model that uses standard MODFLOW parameters it converts them to regular GMS key value approach parameters. The parameters can then be edited using the *Parameters* Dialog. Recharge, evapotranspiration, LPF, and HUF parameters are special in that you can edit the instance and cluster information associated with these types of parameters.

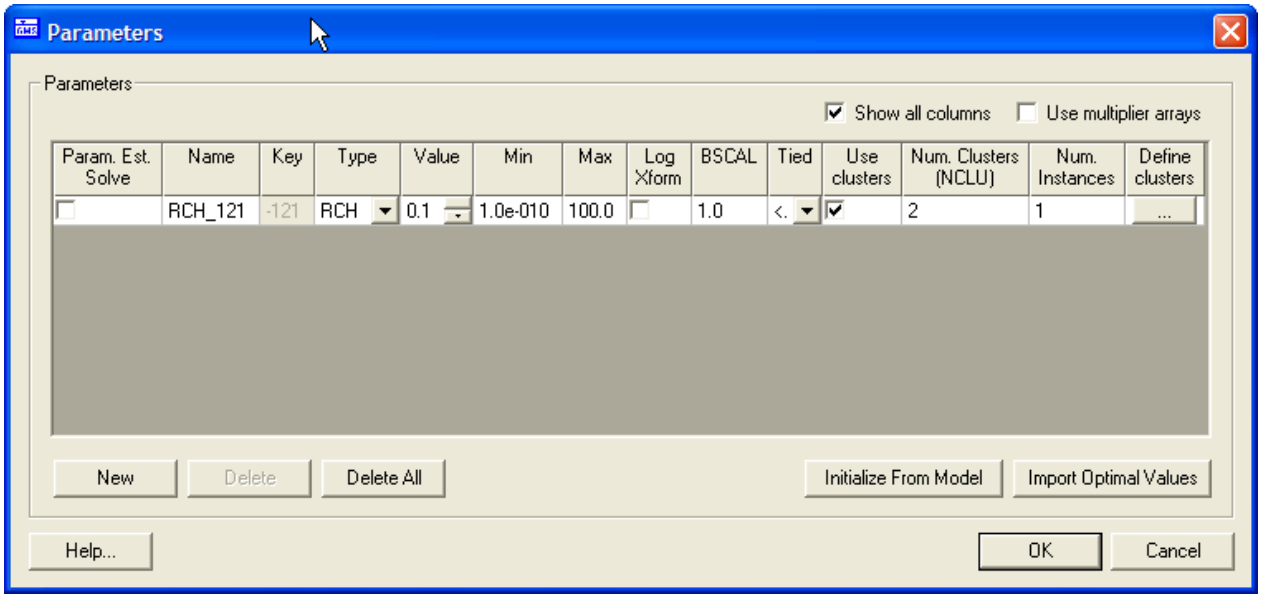
MODFLOW Parameter Clusters Dialog

The *MODFLOW Parameter Clusters* Dialog allows users to define instances for RCH, EVT, LPF, and HUF parameters that use clusters. The inputs in this dialog basically follow the necessary inputs for defining a parameter instance in a package file. You may wish to review the MODFLOW Parameter Clusters and Instances section below.

Array based parameters are defined using clusters. A cluster is a multiplier array, a zone array, and specified zone values where the parameter is applied. Multiplier arrays and zone arrays have associated data sets underneath the three dimensional grid in the Project Explorer.

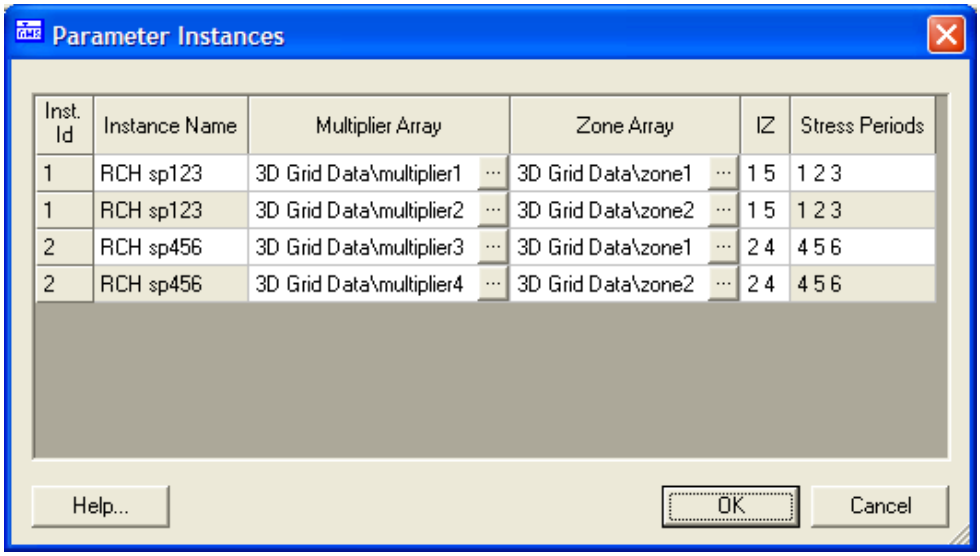
Instances are enabled for RCH and EVT parameters. Instances allow the user to define multiple sets of clusters that are all tied to one parameter. Then the user may use different instances in different stress periods in their MODFLOW model. For more information on parameter instances consult the MODFLOW documentation.

In the MODFLOW Parameters dialog the user selects the **Define clusters** option for a parameter. This will allow the user to specify the number of instances and clusters that are used to define the parameter.



In the *Parameter Instances* dialog the user defines a name for each parameter instance for RCH and EVT parameters. For LPF parameters a level is defined and for HUF parameters a hyrdogeologic unit is defined. Then for each cluster the user selects a multiplier array and a zone array. These are data sets on the 3D Grid. The user must also specify the zone values (IZ) where the parameter is to be applied in the model. If there is more than one IZ value for the cluster then the user enters the numbers with spaces between the entries. So if the parameter is to be applied where the zone array values are 1 and 3 then the user would enter "1 3". The user must also specify the stress periods where an instance is to be used. The numbers entered in the Stress Periods field should also be space delimited. This means if you want an instance to be used in stress periods 1, 2, and 5 then your entry in the Stress Periods field should be "1 2 5".

When the package is written out for MODFLOW the instances will be included in the parameter definition and used for forward runs or for parameter estimation runs.



MODFLOW Parameter Clusters and Instances

For those wanting to follow the standard MODFLOW approach to parameterization, the following explanation may be helpful. You should refer to the MODFLOW documentation for more information.

Definitions

Here is a brief review of the most important terms used when dealing with parameters.

- *Parameter* – Has a name, type, one or more clusters, and zero or more instances.
- *Parval* – The value of the parameter.
- *Cluster* – Has a multiplier array (*Mltarr*), a zone array (*Zonarr*), and zone numbers (*IZ*).
- *Mltarr* – Name of the multiplier array. Parval is multiplied by this to define the value.
- *Zonarr* – Name of the zone array used to define the cells that are associated with a parameter.
- *IZ* – Up to 10 zone numbers (separated by spaces) that define the cells that are associated with a parameter.
- *Instance* – Parameter cluster that could be used for different stress periods in a MODFLOW simulation.

Examples

Below are some examples showing different ways parameters can be used. The examples use parameters to define recharge for a forward run (not doing inverse modeling). The grid in the examples is 5 rows, 5 columns, 1 layer. The examples use the following multiplier and zone arrays.

Multiplier arrays

MULT0001

1.0	1.0	1.0	1.0	1.0
1.0	1.0	1.0	1.0	1.0
1.0	1.0	1.0	1.0	1.0
1.0	1.0	1.0	1.0	1.0
1.0	1.0	1.0	1.0	1.0

MULT0002

0.5	0.5	0.5	0.5	0.5
0.5	0.5	0.5	0.5	0.5
0.5	0.5	0.5	0.5	0.5
0.5	0.5	0.5	0.5	0.5
0.5	0.5	0.5	0.5	0.5

Zone Arrays

ZONE0001

2	2	2	0	0
2	2	2	0	0
2	2	2	0	0
0	0	0	0	0
0	0	0	0	0

ZONE0002

3	3	3	3	3
3	3	3	3	3
4	4	4	4	4
4	4	4	4	4
5	5	5	5	5

ZONE0003

0	0	0	0	0
0	0	6	7	0
0	0	6	7	0
0	0	0	0	0
0	0	0	0	0

Example 1

Steady state. 1 recharge parameter with 1 cluster, 1 instance.

Recharge file	Variables	Explanations
PARAMETER 1	PARAMETER NPRCH	One recharge parameter will be used
3 40	NRCHOP IRCHCB	Apply to highest active cell, save CCF to unit 40
Param1 RCH 0.00005 1	PARNAM PARTYP Parval NCLU	Parameter name, type, value and number of clusters
MULT0001 ZONE0001 2	Mltarr Zonarr IZ	Multiplier array, zone array, and zone number
1 1	INRECH INIRCH	One parameter used in current stress period, INIRCH (ignored)
Param1	Pname	Name of parameter used to define RECH in this stress period

Results

The resulting recharge applied to the top-most active layer would be:

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0	0	0	0	0
0	0	0	0	0

Example 2

Steady state. 2 recharge parameters:

- the first with 1 cluster, 1 instance
- the second with 2 clusters, 1 instance.

Recharge file	Variables	Explanations
PARAMETER 2	PARAMETER NPRCH	Two recharge parameters will be used
3 40	NRCHOP IRCHCB	Apply to highest active cell, save CCF to unit 40
Param1 RCH 0.00005 1	PARNAM PARTYP Parval NCLU	Parameter name, type, value and number of clusters
MULT0001 ZONE0001 2	Mltarr Zonarr IZ	Multiplier array, zone array, and zone number
Param2 RCH 0.00004 2	PARNAM PARTYP Parval NCLU	Parameter name, type, value and number of clusters
MULT0002 ZONE0002 4 5	Mltarr Zonarr IZ	Multiplier array, zone array, and zone numbers
MULT0002 ZONE0003 6 7	Mltarr Zonarr IZ	Multiplier array, zone array, and zone numbers
2 1	INRECH INIRCH	Two parameters used in current stress period, INIRCH (ignored)
Param1	Pname	Name of parameter used to define RECH in this stress period
Param2	Pname	Name of parameter used to define RECH in this stress period

Results

The intermediate results for each parameter cluster are listed below:

Param1

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0	0	0	0	0
0	0	0	0	0

Param2, cluster 1

0	0	0	0	0
0	0	0	0	0
0.00002	0.00002	0.00002	0.00002	0.00002
0.00002	0.00002	0.00002	0.00002	0.00002
0.00002	0.00002	0.00002	0.00002	0.00002

Param2, cluster 2

0	0	0	0	0
0	0	0.00002	0.00002	0
0	0	0.00002	0.00002	0
0	0	0	0	0
0	0	0	0	0

The final result after accumulating each cluster is:

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00007	0.00002	0
0.00007	0.00007	0.00009	0.00004	0.00002
0.00002	0.00002	0.00002	0.00002	0.00002
0.00002	0.00002	0.00002	0.00002	0.00002

Example 3

Transient. 1 recharge parameter with 1 cluster, 2 instances. One instance is used in the first and third stress periods, and the other is used in the second stress period.

Recharge file	Variables	Explanations
`PARAMETER 1`	`PARAMETER NPRCH`	`One recharge parameters will be used`
`3 40`	`NRCHOP IRCHCB`	`Apply to highest active cell, save CCF to unit 40`
`Param1 RCH 0.00005 1 INSTANCES 2`	`PARNAM PARTYP Parval NCLU INSTANCES NUMINST`	`Parameter name, type, value number of clusters, number of instances`
`Instance1`	`INSTNAM`	`Instance name`
`MULT0001 ZONE0001 2`	`Mltarr Zonarr IZ`	`Multiplier array, zone array, and zone number`
`Instance2`	`INSTNAM`	`Instance name`
`MULT0002 ZONE0001 2`	`Mltarr Zonarr IZ`	`Multiplier array, zone array, and zone number`
`1 1`	`INRECH INIRCH`	`One parameters used in current stress period, INIRCH (ignored)`
`Param1 Instance1`	`Pname Iname`	`Name of parameter and name of instance used in this stress period`
`1 1`	`INRECH INIRCH`	`One parameters used in current stress period, INIRCH (ignored)`
`Param1 Instance2`	`Pname Iname`	`Name of parameter and name of instance used in this stress period`
`1 1`	`INRECH INIRCH`	`One parameters used in current stress period, INIRCH (ignored)`
`Param1 Instance1`	`Pname Iname`	`Name of parameter and name of instance used in this stress period`

Results

The intermediate results for each parameter cluster are listed below:

Stress Period 1, Param1, Instance RCH_1_1

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0	0	0	0	0
0	0	0	0	0

Stress Period 2, Param1, Instance RCH_1_2

0.000025	0.000025	0.000025	0	0
0.000025	0.000025	0.000025	0	0
0.000025	0.000025	0.000025	0	0
0	0	0	0	0
0	0	0	0	0

Stress Period 3, Param1, Instance RCH_1_1

0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0.00005	0.00005	0.00005	0	0
0	0	0	0	0
0	0	0	0	0

7.3. Stochastic Modeling

Stochastic Modeling

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

There are two methods in GMS for stochastic modeling using MODFLOW 2000. The first, parameter zonation, uses either a Random Sampling, Latin Hypercube Sampling, or Gaussian Fields to generate the different realizations. The second approach uses indicator simulations generated by T-PROGS.

After the stochastic simulation results are generated, you can view these results using the Project Explorer. You can also refine the results by using the Risk Analysis Wizard or Statistical Analysis of Stochastic solutions.

Stochastic Applications

One approach for dealing with model heterogeneity is stochastic simulations based on multiple equally plausible candidate realizations of the site heterogeneity. Ideally, such an approach would enable the generation of variability in subsurface soil stratigraphy based on interpretable geologic parameters such as lens width, material proportions, juxta-positioning tendencies and anisotropy. Multiple realizations that are conditioned to borehole data provide modelers with a rational approach for dealing with uncertainty associated with site characterization. Stochastic simulations can be applied to regional representations of the aquifer behavior in addition to local scale simulations. Stochastic simulations are particularly well-suited to local scale models since the resulting complex heterogeneity is more representative of actual stratigraphic deposition. This heterogeneity makes realistic and potentially more accurate contaminant transport simulation possible by simulating the preferential flow channels resulting from thin lenses of clays, sands, or other materials. The ultimate result of a stochastic approach is multiple simulations of hydraulic parameters that create a probabilistic solution. Such a solution has more credence and provides a better understanding of actual site conditions.

The ultimate application of T-PROGS is to generate stochastic simulations of soil heterogeneity. These realizations can then be incorporated into MODFLOW simulations as "material sets" in the LPF package or as "HUF data" in the HUF package. (See T-PROGS)

Stochastic Inverse Modeling

Stochastic inverse modeling is a MODFLOW run option that takes each run in a stochastic simulation and performs parameter estimation on the run to find the optimal values based on observation data. This option is very time consuming compared with a regular stochastic simulation and a parameter estimation run because you are doing parameter estimation for each stochastic run times.

Stochastic inverse modeling can be performed only when using material sets or HUF arrays as chosen in the *Stochastic Options* dialog. PEST is the parameter estimation code supported by GMS.

The following occurs during the stochastic inverse process:

1. Run MODFLOW for the stochastic simulation iteration.
2. Set the starting head equal to the resulting heads from the MODFLOW run.
3. Run parameter estimation.
4. Run final MODFLOW (if needed for PEST).
5. Move to next stochastic simulation iteration.

The starting heads are interpolated to decrease the run times during the inverse portion.

Stochastic Options Dialog

You use the stochastic options dialog to select the type of stochastic simulation its major options.

Parameter Randomization

This option uses parameters defined in *Parameters* Dialog.

Material Sets

This option becomes available when there is at least one material set simulation in memory. One method for creating material sets is to use T-PROGS. When this stochastic option is chosen, MODFLOW will be run once for each material set. This option also requires that the LPF package and the material IDs option be chosen.

HUF Sets

This option becomes available when there is at least one HUF set simulation in memory. One method for creating HUF sets is to use T-PROGS. When this stochastic option is chosen, MODFLOW will be run once for each HUF set. This option also requires that the HUF package be chosen.

Only Save Solutions

With this option selected, during each stochastic iteration, changes are made to MODFLOW simulation and only a few files are saved. If this option is not selected, each stochastic iteration results in a new entire set of MODFLOW files being saved.

Using this option requires less disk space, but doesn't allow you to load and rerun individual stochastic iteration simulations.

Using Parameter Zonation With Stochastic Modeling

To create a stochastic MODFLOW simulation using parameter zonation, you follow these steps:

- First define your zones using key values.
- Define parameters that link with your zones.
- Select the **Stochastic Simulation** option from the *Global Options* dialog.
- Select the Parameter Randomization option from the *Stochastic Options* dialog.
- Choose whether you want to use the Random Sampling or Latin Hypercube randomization approaches in the *Parameters* dialog.
- Save and Run your model.
- View the different model results using the Project Explorer.
- Further analyze the results using the Risk Analysis Wizard.

Random Sampling

Random Sampling is the most widely used approach for generating multiple random model simulations. GMS supports both normal and uniform distributions.

A normal distribution can be defined as:

$$f(x) = \frac{1}{\sqrt{2\pi\sigma}}\exp\left(-\frac{(x-u)^2}{2\sigma^2}\right)$$

where σ is the standard deviation, μ is the mean, and x is the value being sampled. A uniform distribution can be defined as:

$$f(x) = \frac{1}{\beta - \alpha}$$

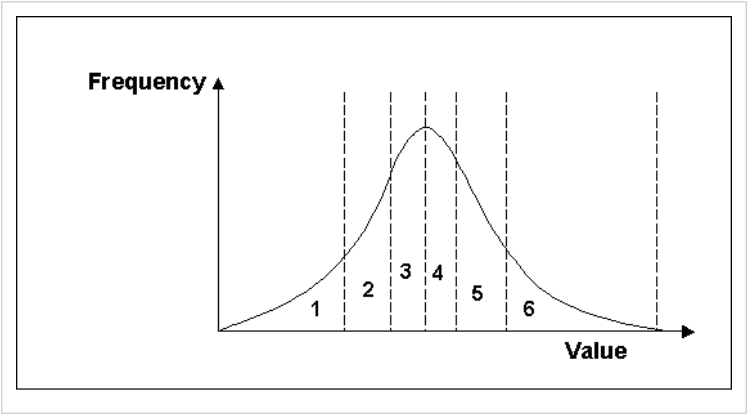
where α and β are the bounds of the parameter value x.

To set up the Random Sampling, you need to specify the mean, standard deviation, and upper and lower bounds for each parameter. Finally, you choose how many realizations you want to generate.

Latin Hypercube

The Latin Hypercube randomization approach is a method that tries to efficiently probe the probability space for each parameter in a simulation in such a way that there is at least one simulation that represents every probability area for each parameter.

First, we specify the number of segments for each parameter. The total probability, defined by a distribution, mean, standard devation, and upper and lower bounds, is divided up into parts with equal probability (area). GMS then generates a random parameter value so that there is one value that lies within each probability segment.



This is repeated in a combinatorial fashion for each parameter so that there are

$$\prod_{i=1}^{n} P_i$$

number of simulations, where n is the number of parameters and P is the number of segments for the ith parameter. For example, if there were three parameters with four, four, and five segments, the number of model runs would be as follows:

$$4 * 4 * 5 = 80$$

Using the Latin Hypercube method has the benefit of needing a fewer number of runs to achieve the same level of confidence than the number required for the Monte Carlo approach because we have guaranteed that the entire probability range will be explored.

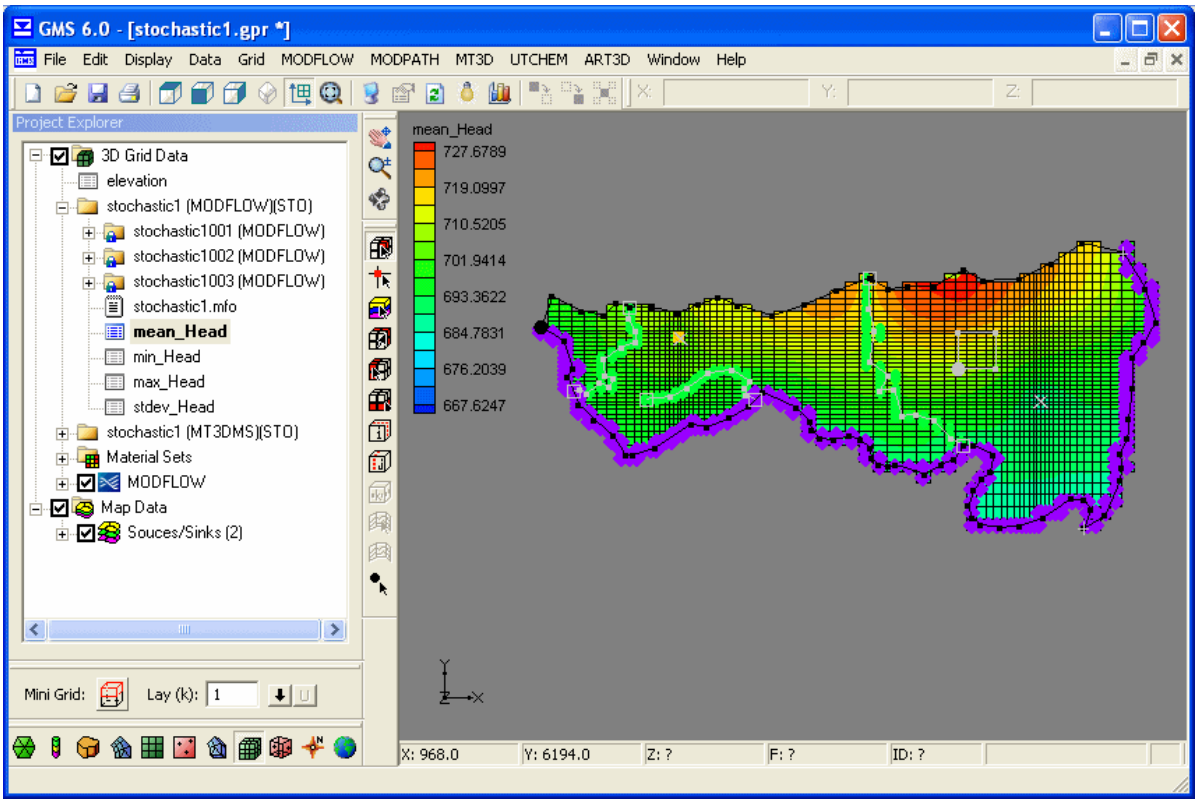
Using Indicator Simulations With Stochastic Modeling

GMS allows you to create a stochastic simulation using output from T-PROGS. This can be in the form of either material sets or HUF data. Use the following steps:

- Generate Material Sets or HUF data using T-PROGS.
- Select the **Stochastic Simulation** from the *Global Options* Dialog.
- Enter the *Stochastic...* dialog from the *MODFLOW* menu.
- Select the Material Sets option and choose the appropriate material set or HUF data simulation.
- Save and Run MODFLOW.
- View the different model results using the Project Explorer.
- Further analyze the results using the Risk Analysis Wizard.
- GMS will generate as many MODFLOW runs as there are material sets.

Statistical Analysis of Stochastic MODFLOW/MT3D solutions

A statistical analysis can be computed on a Stochastic MODFLOW or MT3D solution by right-clicking on a stochastic folder in the Project Explorer and selecting the ***Statistical Analysis*** command. This command creates four data sets for the mean, min, max, and standard deviation. The 3D grid display options can then be used to visualize these data sets.



Gaussian Field Generator

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

GMS includes an interpolation option associated with the 2D scatter point module called Gaussian Sequential Simulation (GSS). This option is used to generate a set of scalar datasets (Gaussian fields) using a Gaussian sequential simulation. This is somewhat similar to indicator kriging or T-PROGS in that it generates a set of equally probable results which exhibit heterogeneity and are conditioned to values at scatter points. However, the resulting arrays are floating point scalar datasets, rather than the integer arrays produced by T-PROGS and indicator kriging.

The results of a GSS can be used in combination with the new Multiplier Array option for parameters. It is now possible to associate one or more scalar datasets with an array-based parameter. When MODFLOW is executed, the parameter starting value is multiplied by the dataset to produce the input array. This makes it possible to use the results of the Gaussian sequential simulation as input for parameter fields for a stochastic (Monte Carlo) simulation.

Gaussian Simulations

The new GSS tool is based on the FIELDGEN code developed by John Doherty. John Doherty describes GSS as follows:

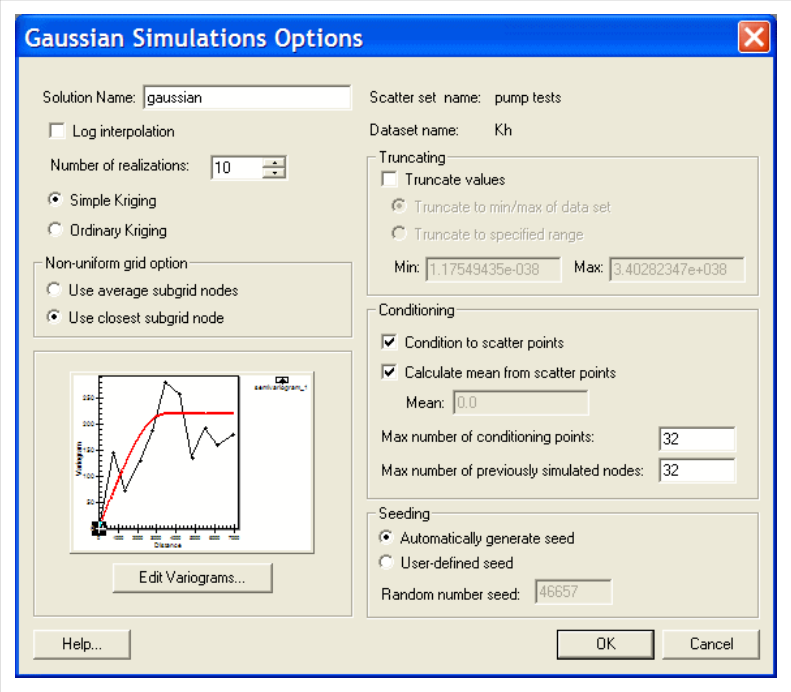
The process of stochastic field generation by sequential simulation is very easy to understand. At each field point an expected field value and a field standard deviation pertaining to that point are first determined. These are calculated through kriging from points to which field values have already been assigned, as well as from points at which conditioning data exists (if available). Using the expected value and standard deviation calculated in this way, a random field value is generated based on the assumption of a Gaussian probability distribution. The field value thus obtained can then be used in generating expected values and standard deviations at other field points at which field generation then takes place in the same way.

GSS is a form of Kriging but it is listed in the GMS interface as a new interpolation scheme. This new option will differ from Kriging in the following ways:

1. GSS uses the FIELDGEN utility developed by John Doherty to perform the interpolation rather than the GSLIB code used by kriging. FIELDGEN is a modified version of the **sgsim** utility in GSLIB so many of the options are quite similar to those used for normal kriging.
2. As is the case with T-PROGS, the user enters the number of desired simulations and FIELDGEN produces N arrays, rather than one array.
3. It can only be used for 2D interpolation and it will only work when interpolating to 3D cell-centered grids.
4. It can work with or without a scatter point set. If a scatter point set is provided, the resulting fields are conditioned to the values at the scatter points. Otherwise the user defines a mean and a variogram and the values are randomly generated.

Gaussian Simulation Options

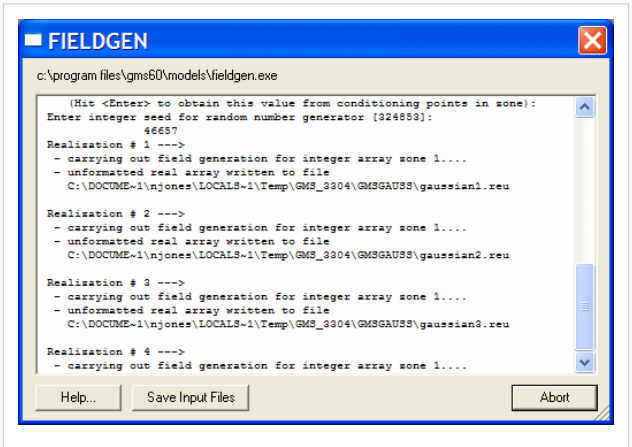
The first step in setting up a GSS is to import a set of scatter points with the values to which you intend to condition your simulation. This step can be skipped if you have no conditioning data. The next step is to select the ***Gaussian Simulation Options*** command in the *Interpolation* menu in the 2D Scatter Point module. This brings up the following dialog:



The **Solution name** at the top is the name that will be applied to the set of Gaussian fields. The **Number of realizations** item is the desired number of Gaussian fields. The original GSLIB code was designed to work with uniform grids (constant cell sizes). The **Non-uniform grid** option controls how the data are converted to a non-uniform grid (if necessary). The **Edit Variogram** button should be selected to set up a model variogram using the GMS variogram editor. A model variogram must be defined whether or not you have scatter points for conditioning.

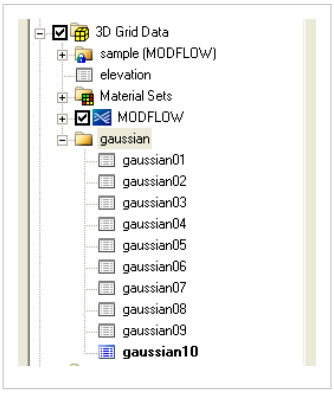
Running the Simulation

Once the GSS options are selected, the next step is to run the simulation. This is accomplished by selecting the ***Run Gaussian Simulation*** command in the *Interpolation* menu. During the simulation, you should see a window displaying the progress of the simulation:

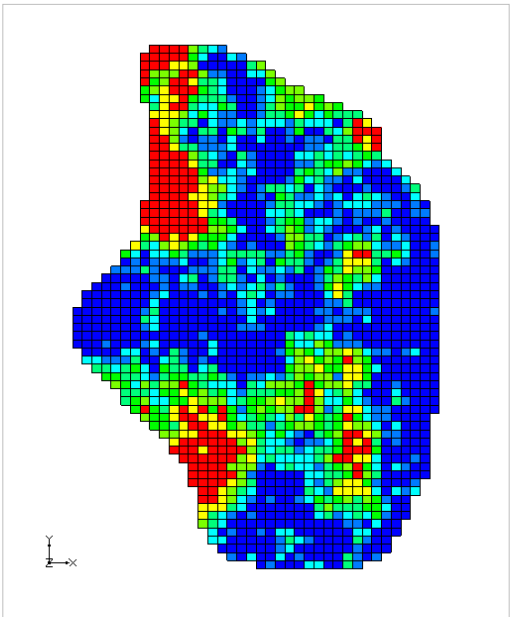


Viewing the Results

Once the simulation is finished, you should see a new folder appear in the Project Explorer window which has the name of the simulation and contains a set of dataset arrays:



Clicking on each data set icon makes it the active dataset for contouring. The dataset properties can be viewed by double-clicking on the icon. The following image represents a sample Gaussian realization:



Risk Analysis Wizard

MODFLOW	
Pre-processing	
Building a MODFLOW Model	
Map to MODFLOW	
Packages Supported in GMS	
Saving a MODFLOW Simulation	
Importing MODFLOW Files	
Unsupported MODFLOW Features	
Post-processing	
MODFLOW Display Options	
MODFLOW Post Processing Viewing Options	
Reading a MODFLOW Simulation	
Tutorials	
MODFLOW Tutorials	
Packages	
Flow:	BCF6, HUF, LPF, UPW
Solvers:	DE4, GMG, NWT, PCG2, PCGN, SAMG, SIP, SOR

Other:	BAS6, CHD1, DRN1, DRT1, EVT1, ETS1, GAGE, GHB1, HFB1, HUF, LAK3, MNW1, MNW2, OUT1, RCH1, RIV1, SFR2, STR1, SUB1, WEL1, UZF1

Calibration, Parameters, Stochastic Modeling
Calibration
Model Calibration
Automated Parameter Estimation
Parameter Estimation Dialog
PEST
Run Options
Observations
Plot Wizard
Calibration Targets
Parameters
Parameters
Parameter Dialog
Pilot Points
Multiplier Arrays for Parameters
Standard MODFLOW Parameters
Stochastic Modeling
Stochastic Modeling
Gaussian Field Generator
Risk Analysis Wizard
T-PROGS

The risk analysis wizard is a tool for refining stochastic modeling results. The Risk Analysis Wizard has two options. You can either perform a Capture Zone Analysis or a Threshold Analysis. Capture zone analysis requires a MODFLOW solution set, but threshold analysis can be performed on any solution set type.

Capture Zone Analysis

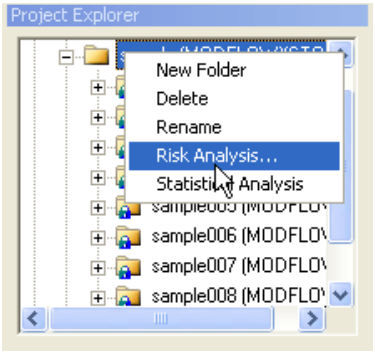
Capture zone analysis includes running MODPATH for each of the MODFLOW solutions to generate a capture zone for each well or zone code group in the MODFLOW model. These capture zones are combined into one probabilistic capture zone

Threshold Analysis

Threshold analysis is similar to capture zone analysis, but threshold analysis can be performed using any type of 3D data set that has been read into the Project Explorer. You first select a simulation set from the Project Explorer. Next, you set up rules for generating a probabilistic threshold dataset. For example, you might be looking at a specific contaminant and want to know what the probability is that the concentration of this contaminant will be above the EPA level for drinking water. You would generate a rule reflecting this limit. After processing all rules for each simulation, GMS creates a probabilistic threshold dataset.

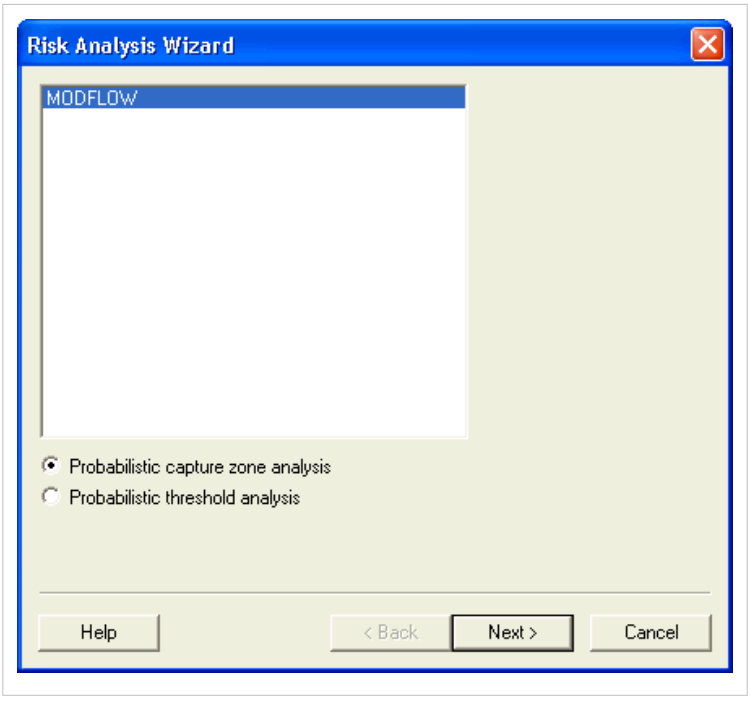
Risk Analysis Wizard Dialogs

The risk analysis wizard is entered through the right-click menu for a folder in the Project Explorer and choose ***Risk Analysis...*** .



Choosing Between Capture Zone and Threshold Analysis

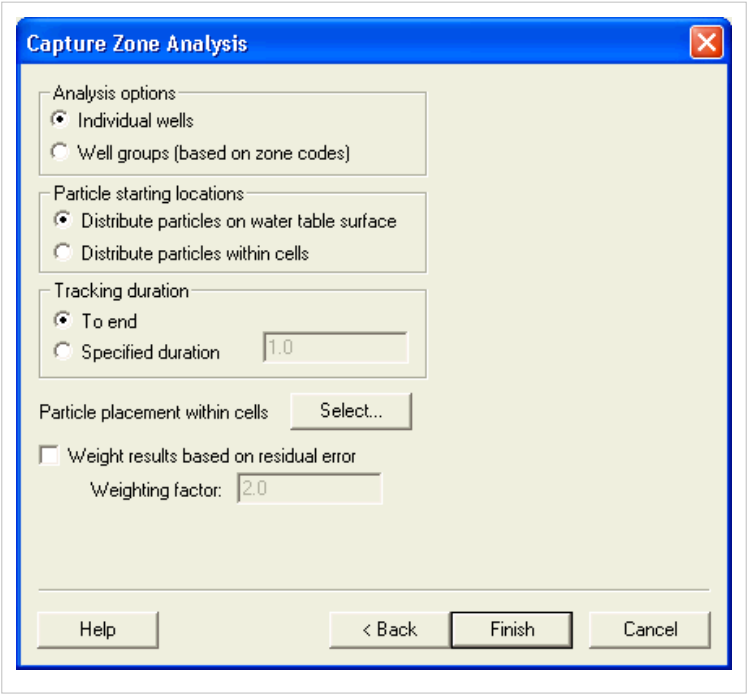
The first step in the risk analysis wizard is to choose what type of simulations you want to process.



The window will list all the solution types as the analysis wizard will only process solutions of the same type. Probabilistic capture zone analysis can only be performed on MODFLOW solutions. Probabilistic threshold analysis can be performed on any solution type.

Capture Zone Analysis Dialog

Capture zone analysis can only be performed on MODFLOW solutions



Analysis Options – Individual wells

Choose this option to create a probabilistic capture zone for each uniquely named well in the MODFLOW model.

Analysis Options – Well groups

Choose this option to create a probabilistic capture zone for each different zone code number.

Particle Starting Locations – Distribute particles on water table surface

Choose this option to distribute particles only on the water table surface. By default, one particle is placed at the *xy* center of the cell at the water table, but you can use the **Particle placement within cells** option to change the number of particles placed on the water table in each cell. This option will create a 2D probabilistic capture zone representing the intersection of the entire 3D probabilistic capture zone with the water table.

Particle Starting Locations - Distribute particles within cells

Choose this option to distribute particles within each cell. By default, one particle is placed at the center of each cell, but you can use the **Particle placement within cells** option to change the number of particles placed on within each cell. This option creates three different probabilistic capture zones. The first represents the 3D capture zone, the other two represent different 2D projections of the 3D capture zone.

Tracking Duration - To end

This option sets MODPATH to move particles through the flow field until they exit the model.

Tracking Duration - Specified duration

This option sets MODPATH to move particles through the flow field until they either exit the model or the time reaches the duration set, which ever comes first.

Particle Placement Within Cells

This option allows you to change the number of particles per cell from the default of one. Increase the number of particles leads to a smoother capture zone, but drastically increases the computation time.

Weight Results Based On Residual Error

In all of the capture zone methods, the algorithms used to synthesize the probability data set can be weighted using observation data. This makes it possible to give more weight to model instances with smaller calibration error when calculating the capture zone probabilities. The weighted head and flow observations can be compared to the computed values to come up with a global error norm, E, for each model run. This error norm can be based on the root mean squared (RMS) error, the sum of the squared weighted residuals, or any other measure selected by the modeler. For capture zone analysis, GMS uses the sum of squared weighted residuals (SSWR) for the error norm. The error norm from each MODFLOW run is used to compute a weight for the given solution using the following equation:

$$w_i = \alpha^{\left[\frac{ME-E_i}{SD}\right]} \quad \text{.....(1)}$$

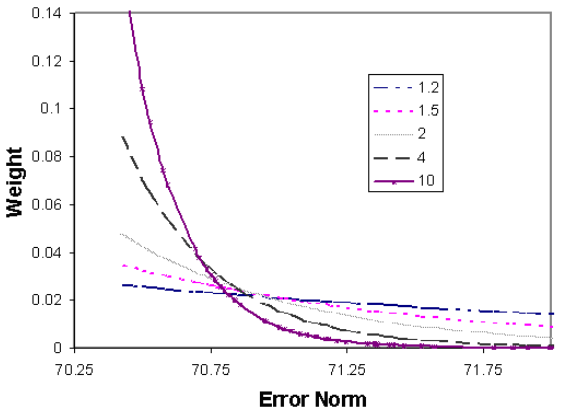
Where W_i is the weight applied to solution i, α is a user-defined factor, ME is the mean of the error values from all solutions, E_{jk} is the error for solution i, and SD is the standard deviation of error values from all solutions. The weights are also normalized as follows

$$w_{final} = \frac{w_i}{\sum w_i} \quad \text{.....(2)}$$

so that the weights sum to unity. Equation 1 was developed to give the greater emphasis to the lower error values and to allow the user to control the relative emphasis given to low vs. high values simply by adjusting the α value. The equation also avoids problems when one of the error values is zero, since a zero error value does not result in an infinite weight. We also wanted the equation to scale the weights according to the data being examined. This is done by subtracting the individual SSWR from the mean error and dividing by the standard deviation.

Equation 1 centers the weights on the mean error. The relative weight given to values differing from the mean is biased by the factor. This makes it possible to bias the resulting weight using knowledge of the site and the quality of the observation data.

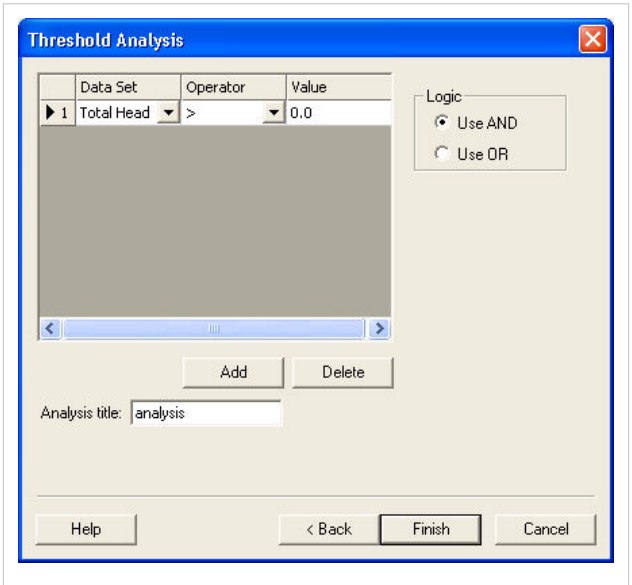
The figure below shows how the α factor in Equation 1 affects the weight applied to a given error. An α factor of 1.2 makes the contribution of each SSWR almost linear, whereas an α factor of ten gives most of the weight to the lowest 5-10 percent while discounting the other error values. We typically use an α value of 2.0.



Weight versus Error Norm Plot

Threshold Analysis Dialog

Threshold analysis can be performed on any collection of 3D datasets. The threshold analysis dataset is created by using rules. For each rule, you select a dataset (only applies to solutions with multiple data sets.), the greater or less than sign and a value. You can have as many rules as you want. The AND and OR logic options allow you to use: 1st rule AND 2nd rule AND 3rd rule; or 1st rule OR 2nd rule OR 3rd rule. Threshold analysis creates one 3D dataset.



8. Importing/Exporting Data

Importing Non-native GMS Files

GMS can import many files generated by other software in their native format. The files that can be imported to GMS are shown in the tables below. Each file type is identified by the file extension. The file filter corresponding to the desired extension should be selected in the *Open File* dialog.

Text Import Wizard

In addition to the file types listed below, several other types of data can be imported via the *Text Import Wizard*. Refer to the section on the *Text Import Wizard* for more information.

File Type	File Ext	Description
Model Super Files	*.mfs, *.rsp, *.mts, *.rts, *.sms, *.uts, *.fws, *.ats, *.sps	GMS uses super files to help organize model files, which can be numerous. Opening the super file will import the entire model and all associated model files. This will NOT open any GMS specific data, such as Map data or TIN data etc. The file models associated with each super file extension are as follows: *.mfs = MODFLOW, *.rsp = MODPATH, *.mts = MT3DMS, *.rts = RT3D, *.sms = SEAM3D, *.uts = UTCHEM, *.fws = FEMWATER, *.ats = ART3D, *.sps = SEEP2D
Text files	*.txt	Text files where the data is in columns, as you might export from a spreadsheet. This brings up the GMS Text Import Wizard. Many different types of data can be brought in to GMS by using the Text Import Wizard.
DXF/DWG	*.dxf, *.dwg	Vector drawing data used for background display or for conversion to feature objects.
JPEG - TIFF	*.jpg, *.tif	Raster image files used for background display or for texture mapping to a surface.
Shapefiles	*.shp	ArcView shapefiles.
DEM / Grid	*.asc, *.ddf, *.ggd, *.dem	ASCII 2D grid exported from Arc/Info or ArcView, ASCII 2D grid exported from GRASS
Surfer Grid	*.grd	ASCII 2D grid exported from Surfer.

Text Import Wizard

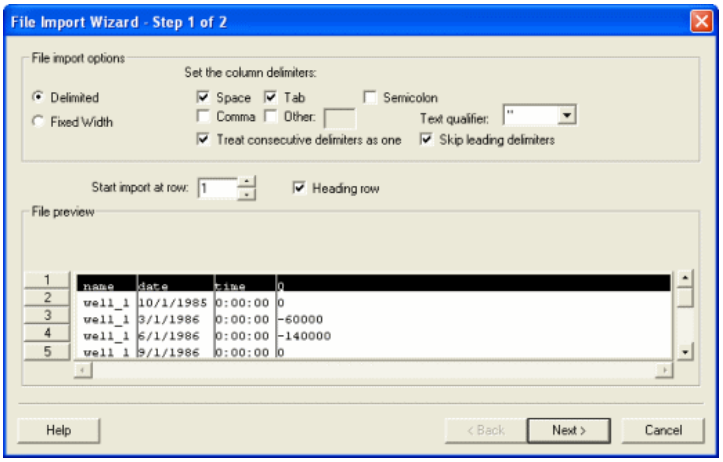
GMS can import many files generated by other software in their native format. Refer to Importing Non-native GMS Files for a list. For files that are not included in the list, GMS provides the *Text Import Wizard*.

The *Text Import Wizard* enables users to import many different types of data into GMS. The Text Import Wizard is initialized by selecting a *.txt file in the ***Open*** command from the *File* menu. The wizard has two steps.

Step 1 – Delimiting Columns

The first step in the wizard allows users to delimit the data into columns. Two options exist to delimit the data: **Delimited**, **Fixed Width**.

For the Delimited option, typical delimiters are included as well as an option for users to specify a delimiter. Columns can also be specified with a fixed width by clicking on the ruler bar or the window with the data. Break lines can be dragged, and they can be deleted by double-clicking on the break line or dragging them off the screen. The user can specify the starting row the data will be imported at. If your data has a row of headings, you can indicate such and GMS will use the headings in the next step to determine what kind of data each column represents.

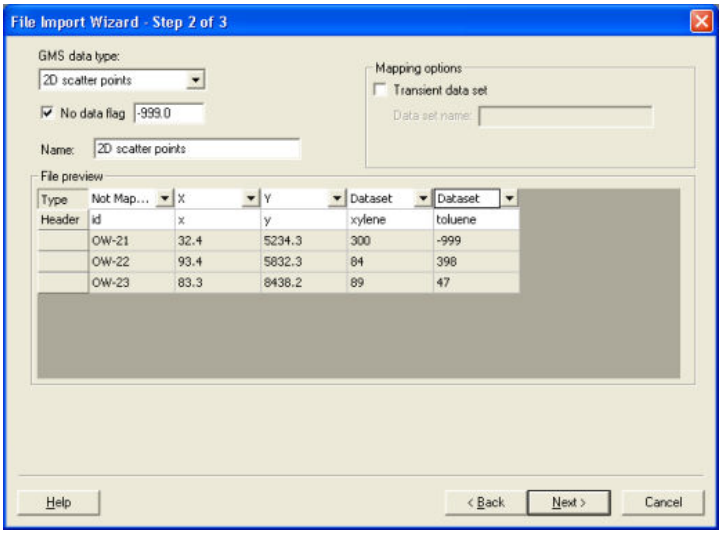


Step 1 File import options of the Text Import Wizard.

Step 2 – Assigning Column Types

The first 20 lines of the file are displayed in a spreadsheet according to the file outline specified in step 1. This step lets you pick what kind of data you are importing (see Supported File Formats). A "no data flag" can be specified for the file. This is a number that, when encountered in the file, tells GMS to mark the value as "NULL" or "no data".

The data in the columns are identified by selecting the type in the combo box at the top of each column in the spreadsheet. If a row of headings exists, GMS will automatically select the proper type if it recognizes the heading. Otherwise they are Not Mapped by default. The available column types changes depending on the GMS data type selected. Certain column types must be mapped for each file format before the user can progress to the next step in the wizard. The name of each column is changed by editing the Header cell.



Step 2 (File Format) of the Text Import Wizard.

After the data have been imported, the coordinate transformation tools can be used to transform and translate the data.

Text Import Wizard Supported File Formats

The Text Import Wizard can import various types of data as described in the tables below.

2D scatter points

This will create new scatter points in a new 2D scatter set. Optionally the imported points can be filtered using the filter options available by selecting the **Filter Options** button.

Field	Type	Required	Comments
X	Number	yes	
Y	Number	yes	
Label	Text	no	
Data Set	Number	no	Any number of steady-state or transient data sets can be imported. When importing transient data sets, a data set name and a time step must be specified for each transient column. Names and time steps are entered into the header row, separated by a vertical slash using the following format: "Name|Timestep". For example, "Dataset1|January 1, 1900". All time steps of a transient data set should be given the same name.

Examples

These examples can be copied and pasted into GMS.

Steady state

```
"id"      "x"      "y"      "xylene"      "toluene 0.0"      "toluene 2.0"
"OW-21"      32.4      74.3      300      999      999
"OW-22"      93.4      32.3      84      398      401
"OW-23"      83.3      48.2      89      47      52
"OW-24"      46.8      29.1      207      147      134
```

Transient

```
"id"      "x"      "y"      "xylene|01/01/2000 00:00"      "xylene|01/01/2000 12:00"      "xylene|01/02/2000 00:00"
"OW-21"      32.4      74.3      300      87      276
"OW-22"      93.4      32.3      84      310      187
"OW-23"      83.3      48.2      89      148      198
"OW-24"      46.8      29.1      207      147      134
"OW-25"      40.3      50.1      158      213      32
"OW-26"      60.7      52.0      27      41      300
"OW-27"      78.7      72.9      264      275      206
```

3D scatter points

This will create new scatter points in a new 3D scatter set.

Field	Type	Required	Comments
X	Number	yes	
Y	Number	yes	
Z	Number	yes	
Label	Text	no	
Material ID	Number	no	
Data Set	Number	no	Any number of steady-state or transient data sets can be imported. When importing transient data sets, a data set name and a time step must be specified for each transient column. Names and time steps are entered into the header row, separated by a vertical slash using the following format: "Name|Timestep". For example, "Dataset1|January 1, 1900". All time steps of a transient data set should be given the same name.

Examples

These examples can be copied and pasted into GMS.

Steady state

```
"id"      "x"      "y"      "z"      "xylene"      "toluene 0.0"      "toluene 2.0"
"OW-21"      32.4      74.3      10.1      300      87      999
"OW-22"      93.4      32.3      25.7      84      310      187
"OW-23"      83.3      48.2      5.3      89      148      198
"OW-24"      46.8      29.1      -42.2      207      147      134
```

Transient

```
"id"      "x"      "y"      "z"      "xylene|01/01/2000 00:00"      "xylene|01/01/2000 12:00"      "xylene|01/02/2000 00:00"
"OW-21"      32.4      74.3      10.1      300      387      276
"OW-22"      93.4      32.3      25.7      84      310      187
"OW-23"      83.3      48.2      5.3      89      148      198
"OW-24"      46.8      29.1      -42.2      207      147      134
```

Observation data

This creates new feature points in the existing, active coverage. The active coverage must exist and have a steady state observation attribute (like "Head") defined.

Field	Type	Required	Comments
Point Name	Text	no	
X	Number	yes	
Y	Number	yes	
Z	Number	no	
Layer	Number	no	The 3D grid layer the point is associated with.
Measurement	Text	no	Measurement name. Multiple measurements allowed.
Interval	Number	no	Use Interval and Confidence, or Std. Dev. but not both.
Confidence	Number	no	Use Interval and Confidence, or Std. Dev. but not both.
Standard Deviation	Number	no	Use Interval and Confidence, or Std. Dev. but not both.

Example

```
"id"      "x"      "y"      "z"      "lay"      "hd"      "int"      "conf"
"OBS_Q5"      23.3      44.2      32.2      1      567.5      1.2      95
"OBS_Q6"      83.3      84.3      32.2      1      555.3      1.4      90
"OBS_Q7"      85.3      39.3      33.2      1      999      0      0
```

Transient observation data

This adds time series curves to existing points in an existing observation coverage. The targeted coverage must be the active coverage.

The existing points and the data being imported are matched by **Name** or **ID**. If both **Name** and **ID** are specified then **ID** will take precedence.

Both **Date** and **Time** can be specified although data can be imported with only **Date** or only **Time** specified (at least one of these fields must be specified). If only the **Date** is specified then the time is set to 12:00:00 AM. If only **Time** is specified then it is assumed that the user is not using date/time format and the time is treated as a relative time.

Field	Type	Required	Comments
Name	Text	no	Either Name or ID must be specified
ID	Text	no	Either Name or ID must be specified
Date	Date	no	Either Date or Time or both must be specified
Time	Time	no	Either Date or Time or both must be specified

Example

```
"id"      "date"      "time"      "head"
"OBS_Q5"      12/3/1999      18:00:00      238.5
"OBS_Q5"      1/30/2000      07:38:25      834.7
"OBS_Q6"      3/27/2000      18:00:00      878.3
"OBS_Q6"      12/3/1999      18:00:00      733.2
```

Well data

This creates new feature points in an existing coverage. The point types are set to "well" and the flow rate and other data are assigned to the points. A coverage with the "Wells" attribute must already exist and be the active coverage.

Field	Type	Required	Comments
Name	Text	no	
X	Number	yes	
Y	Number	yes	
Z	Number	no	Elevation of top of well.
Depth	Number	no	Depth to top of screen.
Screen Length	Number	no	
Flow Rate	Number	no	Positive for injection, negative for extraction.

Example

```
"id"      "x"      "y"      "z"      "depth"      "len"      "Q"
"well_1"      123.5      843.2      62.0      51.4      6.0      120.5
"well_2"      134.2      789.4      62.8      60.7      5.4      100.3
"well_3"      147.2      678.1      61.4      110      10.0      300.5
```

Pumping data

This adds time series curves to existing points in an existing coverage containing wells. The targeted coverage must be the active coverage. The existing points and the data being imported are matched by name.

Field	Type	Required	Comments
Well Name	Text	yes	Must match an existing well, or else data is ignored.
Date	Date	No	Either Date or Time or both must be specified.
Time	Time	No	Either Date or Time or both must be specified.
Flow Rate	Number	yes	Positive for injection, negative for extraction.
Depth	Number	no	Depth to top of screen.
Screen Length	Number	no	

Example

```
"id"      "date"      "time"      "Q"
"well_1"      12/3/1999      18:00:00      625.0
"well_1"      1/30/2000      07:38:25      0
"well_2"      12/3/1999      18:00:00      0
"well_2"      3/3/2000      14:48:32      100.0
```

Borehole data

This creates new boreholes. Optionally, a borehole folder name can be specified to organize the imported boreholes.

Field	Type	Required	Comments
Borehole Name	Text	yes	
X	Number	yes	X location of the contact.
Y	Number	yes	Y location of the contact.
Z	Number	yes	Z location of the contact.
HGU ID	Number	yes	HGU Material below the contact
Material ID	Number	no	Material below the contact.
Horizon ID	Number	no	

```
"id"      "x"      "y"      "z"      "material"      "horizon"
"borehole_1"      0.0      0.0      0.0      3      1
"borehole_1"      0.0      0.0      10.0      1      2
"borehole_1"      0.0      0.0      15.0      6      4
"borehole_2"      5.0      0.0      1.0      2      3
"borehole_2"      5.0      0.0      6.0      2      4
```

Borehole sample data

This creates new sample data on existing boreholes, or new sample data on new boreholes if no boreholes with matching names exist. All existing sample data is replaced with that being imported.

Field	Type	Required	Comments
Name	Text	yes	New Borehole Name
X	Number	yes	X location of the borehole.
Y	Number	yes	Y location of the borehole.
Z	Number	yes	Z location of the borehole.
Data Set	Number	yes	

```
"id"      "x"      "y"      "z"      "data"
"borehole_1"      0.0      0.0      0.0      495
"borehole_1"      0.0      0.0      10.0      484
"borehole_1"      0.0      0.0      15.0      454
"borehole_2"      5.0      0.0      1.0      259
"borehole_2"      5.0      0.0      6.0      248
```

Transient point data

This creates new time series curve data on existing feature points similar to the "Pumping data" option above. This option is more generic than the "Pumping data" option, however, because it will work with feature points of any type and any attribute (that can be transient) of that feature point.

The existing points and the data being imported are matched by **Name** or **ID** or **X** and **Y**. The order of precedence for these fields is **ID**, **Name**, then **X** and **Y**.

Both **Date** and **Time** can be specified although data can be imported with only **Date** or only **Time** specified (at least one of these fields must be specified). If only the **Date** is specified then the time is set to 12:00:00 AM. If only **Time** is specified then it is assumed that the user is not using date/time format and the time is treated as a relative time.

Data for more than one attribute can be imported at the same time. The attributes of the active coverage are listed as available fields that can be mapped.

Field	Type	Required	Comments
Name	Text	No	Name or ID or X and Y must be specified.
ID	Number	No	Name or ID or X and Y must be specified.
Date	Date	No	Either Date or Time or both must be specified.
Time	Time	No	Either Date or Time or both must be specified.
X	Number	No	Name or ID or X and Y must be specified.
Y	Number	No	Name or ID or X and Y must be specified.
[Attribute1...]	Number	No	The value of the selected attribute at the given date/time.
[AttributeN]	Number	No	The value of the selected attribute at the given date/time.

```
"id"      "date"      "time"      "x"      "y"      "Flow rate"
"well_1"      12/3/1999      18:00:00      123.5      843.2      625
"well_1"      1/30/2000      07:38:25      123.5      843.2      0
"well_2"      3/27/2000      18:00:00      134.2      789.4      0
"well_2"      12/3/1999      18:00:00      134.2      789.4      100.0
```

Transient node data

Similar to Transient point data but for nodes. Since nodes don't have names they must be matched by ID or XY location.

Field	Type	Required	Comments
ID	Number	No	Either ID or X and Y must be specified.
Date	Date	No	Either Date or Time or both must be specified.
Time	Time	No	Either Date or Time or both must be specified.
X	Number	No	Either ID or X and Y must be specified.
Y	Number	No	Either ID or X and Y must be specified.
[Attribute1...]	Number	No	The value of the selected attribute at the given date/time.
[AttributeN]	Number	No	The value of the selected attribute at the given date/time.

Transient arc data

Similar to Transient point data but for arcs. Since arcs don't have a single XY location they must be matched by ID or Name.

Field	Type	Required	Comments
Name	Text	No	Either Name or ID must be specified.
ID	Number	No	Either Name or ID must be specified.
Date	Date	No	Either Date or Time or both must be specified.
Time	Time	No	Either Date or Time or both must be specified.
[Attribute1...]	Number	No	The value of the selected attribute at the given date/time.
[AttributeN]	Number	No	The value of the selected attribute at the given date/time.

Transient polygon data

Similar to Transient point data but for polygons. Since polygons don't have a single XY location they must be matched by ID or Name.

Field	Type	Required	Comments
Name	Text	No	Name or ID must be specified.
ID	Number	No	Name or ID must be specified.
Date	Date	No	Either Date or Time or both must be specified.
Time	Time	No	Either Date or Time or both must be specified.
[Attribute1...]	Number	No	The value of the selected attribute at the given date/time.
[AttributeN]	Number	No	The value of the selected attribute at the given date/time.

Importing/Exporting CAD Data

Importing

GMS can import DWG or DXF files via the ***File|Open*** command. If there is already CAD data in memory, GMS will replace the existing data with the data being imported. Currently, GMS cannot merge the incoming data with the data in memory.

Exporting

GMS data can be exported to a DWG or DXF file that can then be read into a CAD software package. If there is CAD data in memory when a GMS project is saved, GMS creates a new DWG file from the CAD data. The file is put in the same folder with the other project files and named using the project prefix.

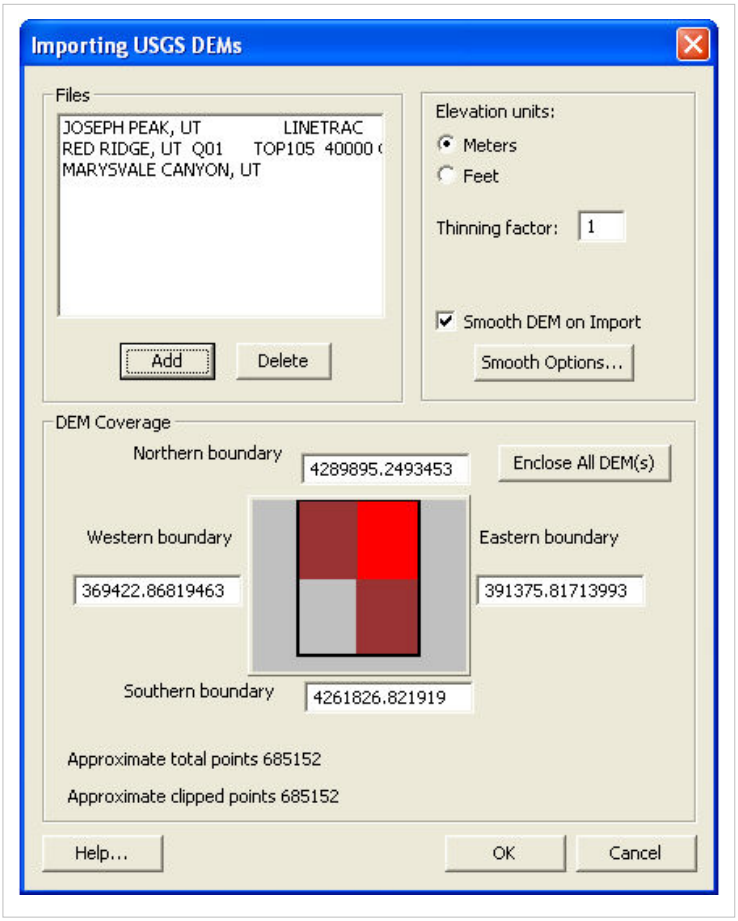
Any CAD data in memory can be exported by right-clicking the CAD folder in the Project Explorer and selecting the ***Export*** command. The *DWG/DXF Filename* dialog will appear, and you can select either the DWG of DXF file types to save the CAD data. GMS objects must first be converted to CAD data before CAD data can be exported. To convert GMS data to CAD data, use the ***Convert To CAD*** command in the *Display* Menu.

Starting at GMS 9.0, multiple CAD files can be included in the project at once. GMS does not write to these files when the GMS project is saved, it merely links to the CAD files.

CAD Properties

The CAD Properties dialog displays data about the CAD file such as: path on disk, extents in xy, and file version.

Importing USGS DEMs



The *Import USGS DEMs* dialog is used to examine the limits of DEM files as well as defining a clipping boundary to eliminate regions outside the area of interest prior to actually reading the elevations in. The **Add** button is used to add a new file to the list of files (you can now select multiple DEM files at the same time rather than adding them one at a time, but they must all be of the same format) that will be read. The standard file opening dialog appears from which you may select DEM files. Once the file(s) are added to the list a bounding rectangle is displayed in the small graphics window in the center of the dialog. As additional files are added the graphics region is updated with new rectangles in order to provide an understanding of where DEMs are located in relation to one another.

A small black rectangle is displayed in the central graphics window. Only elevation points inside this rectangular region will be read in when hitting OK from this dialog. This boundary rectangle can be modified in three different ways.

- **DRAGGING –** Using the mouse you can click near one of the four edges of the bounding rectangle and drag it to a new location. If you click near a corner both edges will be adjusted. If you click in the center of the rectangle then the entire rectangle can be translated to a new location. As you drag edges to new locations their corresponding values are automatically updated in the Edit Fields.
- **EDIT FIELDS –** Any one of the western, eastern, northern, or southern boundaries can be explicitly set by changing the values in their corresponding edit fields. As new values are entered the display in the small graphics window can be updated by tabbing or by clicking the cursor outside the current edit field.
- **ENCLOSE ALL DEMs –** This button can be used to force the edges of the bounding rectangle to correspond to the limits of the DEM files which have been added to this point. By default when a new DEM file is added the bounding rectangle is adjusted to enclose all DEMs.

The thinning factor can be used to reduce the number of elevation points read. A thinning factor of 2 means that every other row and column would be read, reducing the number of total points by a factor of 4. A factor of three means that every third row and column would be read reducing the total by a factor of 9, etc.

The elevation units toggle can be used to specify whether imported DEM points have meter or feet for units of elevation. If a DEMs base elevation units are feet and the toggle specified meters, all elevations are converted when reading. This is particularly important when trying to read two adjacent DEMs with different base elevation units. Also see about transforming coordinate systems.

NOTE: This option does not change the base planimetric units of the DEM and you should make the elevation units consistent with the planimetric units in order to ensure that slopes are computed properly when computing basin geometric parameters.

At the bottom of this dialog the total number of DEM points from all DEM files which have been added and the approximate number of points inside the bounding rectangle are displayed. These numbers can be used to determine how many points your system is capable of reading. For example, each DEM point requires 5 bytes of memory, so that if you read an entire 1:250,000 DEM with about 1.4 million points, 1.4 meg * 5 bytes = 7 meg of memory would be required (in addition to whatever other memory being used by GMS). This means that you would need at least 8 meg of RAM, or some type of virtual memory capabilities would be required to read in the entire DEM file.

This same dialog is used for all five types of DEMs supported in the GMS import options: USGS, ARC/INFO®, GRASS, DTED, and SDTS.

DEMs in Different UTM Zones

Occasionally two adjacent USGS DEM files will be read in but do not appear adjacent in the import dialog. This occurs because while the two DEMs are adjacent, they lie in different UTM zones. X coordinates within UTM zones repeat and therefore the DEMs do not lie adjacent to one another as they should. WMS does not contain the utility functions necessary to transform a DEM from one UTM coordinate zone to another. However, GMS, ARC/INFO® and possibly other GIS software can be used to convert from one zone to another.

Importing/Exporting GIS Grids

GMS includes an option to import and export grid files from/to either the GRASS or ARC/INFO (ArcView) geographic information systems formats.

Importing

Grids are imported using the ***Open*** command in the *File* menu.

With GRASS and ARC/INFO grid files, the grid and the attributes are saved to a single file. When a grid file is imported, GMS constructs the grid, and then reads in the attribute matrix as a dataset.

When a grid in GRASS or ARC/INFO contains multiple attributes (datasets), each attribute is written to a separate grid file. In such cases, it is possible to read in one instance of the grid file to create a grid with a single dataset, and then read in the other grids as extra datasets on the first grid. This can be accomplished as follows:

1. Select the ***Open*** command from the *File* menu.
2. Choose either the GRASS or ARC/INFO file filter.
3. Select the grid file.
4. Select the **OK** button.

At this point, the first grid file is imported resulting in a grid with a single dataset. For the additional datasets, steps 1-5 are repeated but now GMS will detect that there is already a 2D grid in memory that matches the grid being imported and will ask whether you want to replace the existing grid, or add the new grid as a data set of the existing grid.

Exporting

Grids can be exported from GMS in the GRASS or ARC/INFO format by right-clicking on the grid in the Project Explorer and selecting the ***Export*** menu command. Only grids with equal row heights and equal column widths can be exported. When a grid is exported, the grid and the active dataset are written to the GIS file. If multiple datasets are associated with the grid, each dataset should be made the active dataset and the grid should be exported repeatedly so that each dataset is written to a separate grid file.

Importing/Exporting Shapefiles

The data model used for feature objects (points, nodes, vertices, arcs, polygons) was patterned after the vector GIS data model used in Arc/Info and ArcView. As a result, feature objects can be imported from Arc/Info or ArcView into GMS or exported from GMS to either Arc/Info or ArcView using shapefiles. A shapefile is a binary file used to store points, arcs, and polygons. Each object type is typically saved to a separate shapefile.

When a shapefile is saved from ArcView or Arc/Info, three files are saved. The files are described in the following table. When the shapefile (*.shp) is imported to GMS, the database file (*.dbf) is automatically imported at the same time.

File Extension	Decription
*.shp	This file contains the geometry of the points, lines, or polygons.
*.dbf	This is a relational database file. The properties of the feature objects are stored in this file.
*.shx	This is an index file. It is ignored by GMS.

Importing Shapefiles

Shapefiles are imported to GMS using ***Open*** command in the *File* menu and selecting the *.shp extension. A dialog comes up that prompts the user whether to import the shapefile as a scatter set or feature objects.

If the scatter set option is selected and the file must is a point file, the points are imported. If the option selected is Feature Objects, the *Map Shapefile Properties* Dialog appears. The list of coverage attributes in the top right portion of the dialog are the properties for the current GMS coverage type. The list of database fields in the top left portion of the dialog are the properties that GMS found in the specified shapefile. The bottom portion of the dialog shows how the shapefile properties will be mapped to the GMS coverage properties. GMS will attempt to automatically set up the mapping by looking for shapefile attribute names that match the names of the GMS coverage properties. The user can manually change the attribute mapping using the **Map** and **Unmap** buttons. Database fields that are not mapped will be ignored by GMS.

The order for importing is first polygons, then arcs, then points. For example, when importing both an arc and a node shapefile, it is important to import the arc shapefile first. The properties that a node can have depend on the arcs it is attached to so GMS must find the arcs attached to nodes being imported. The points on the arcs need to be nodes and not vertices for GMS to match them up. That is, the points on the arcs that correspond with the points being imported in the point shapefile must be nodes, not vertices. Even if they're in the exact same location, GMS won't match them up if they're vertices, only if they're nodes.

With the current version of GMS, only steady state data can be imported from a shapefile. Transient properties are not supported. Transient well data can be imported using a Pumping Well Data file. Transient observation point data can be imported using a Tabular Obersvation Point file.

Exporting Shapefiles

GMS feature objects are saved to shapefiles by right-clicking a coverage in the Project Explorer and using the ***Export*** command. Saving shapefiles brings up the Export Shapefile dialog. The *Export Shapefile* dialog shows the path and filename where the shapefile will be saved. In GMS, a single coverage can contain points, nodes, arcs and polygons. Shapefiles, however, can only contain one type of attribute. Thus, GMS can export up to three shapefiles depending on the type of objects in the GMS coverage. For example, if the GMS coverage contains arcs, points and nodes, the user can export the arcs to one shapefile and the points and nodes to a separate point shapefile. The user can control which shapefiles GMS will create when more than one is possible. The names for the shapefiles come from the prefix listed at the top of the dialog, appended with "_pts" for the point shapefile, "_arcs" for the arc shapefile, and "_polys" for the polygon shapefile.

The Export Shapefile dialog allows the user to specify the format of fields containing floating point numbers in the shapefile. The total width of the field and the number of digits to the right of the decimal can be specified. All float fields in the shapefile will be formatted according to these settings.

With the current version of GMS, only steady state data can be exported to a shapefile. Transient properties are not saved to the file.

The table of attributes that is exported matches the table that is displayed in the properties dialog when editing feature objects.

Currently, you can only export one coverage at a time to shapefiles.

Shapefile Properties

The tables below indicate how GMS imports and exports shapefile properties. A "type" field must exist as an attribute when importing so GMS knows what kind of attribute to assign to the object. If the "type" field is not found, GMS will import the objects as generic objects without properties and the Map Shapefile Properties dialog will not appear.

A single shapefile may contain objects with different attribute types. Different attribute types require different fields, as shown in the tables. The X's in the table indicate which fields correspond with which attribute types. GMS writes -999 to empty cells when exporting because shapefiles have no way of flagging "null" or "no data" fields. If a field is not required based on the attribute types of the objects to be exported, the field will not be included.

GMS only supports importing of steady state data (one moment in time) from shapefiles.

MODFLOW/MT3D Local Sources Sinks Coverage

For MODFLOW/MT3D local source/sink coverages, if a node is attached to more than one arc, it can have more than one attribute type. When GMS exports these nodes, it exports one node for every attribute type the node has. So the shapefile might end up with several points in the same location, each with different properties. When importing, GMS will resolve all nodes found in the same location to one node and assign the node the union of the properties of all the nodes.

Note that the following options only work for steady state data. Transient well data can be imported using the Pumping Well Data File formats.

Type	Z	Cond	Elev	Stage	Flux	Screentop	Screenshot	Beginlayer	Endlayer
Generic	X								
Shead	X		X					X	X
Sconc	X							X	X
Ghead	X	X	X					X	X
Drn	X	X	X					X	X
Riv	X	X	X	X				X	X
Well	X				X	X	X	X	X

Point Properties For MODFLOW/MT3D Local Source/sink Coverage.

Type	Z	Elev	Stage	Topelev	Botelev
Shead	X	X			
Sconc	X				
Ghead	X	X			
Drn	X	X			
Riv	X	X	X		
Strm	X		X	X	X

Node Properties For MODFLOW/MT3D Local Source/sink Coverage.

Type	Cond	Elev	Stage	Beginlayer	Endlayer
Generic					
Shead		X		X	X
Sconc				X	X
Ghead	X	X		X	X
Drn	X	X		X	X
Riv	X	X	X	X	X

Polygon Properties for MODFLOW/MT3D Local Source/sink Coverage.

MODFLOW/MT3D Areal Coverage

The fields exported for MODFLOW/MT3D layer coverages depend on what properties are defined for the polygons. They also depend on which flow package (LPF, BCF or HUF) is selected in the *Coverage Properties* dialog. Similarly when importing GMS looks for the appropriate fields based on the selected flow package.

The following table lists the attributes that go with each flow package.

Shapefile Field	Meaning	LPF	BCF	HUF
Type	The attribute type ("Layer")	X	X	X
Topelev	Top Elevation	X	X	X
Botelev	Bottom Elevation	X	X	X
Trans	Transmissivity		X	
Kh	Horizontal hydraulic conductivity	X	X	
KV	Vertical hydraulic conductivity	X		
Leak	Leakance		X	
Hanis	Horizontal Anisotropy	X		
Vanis	Vertical Anisotropy	X		
Specstore	Specific storage	X		
Specyield	Specific yield	X		
Pstore	Primary storage coefficient		X	
Sstore	Secondary storage coefficient		X	
Wetdry	Wet/dry flag	X	X	X
Zonecode	Zone code	X	X	X
Porosity	Porosity	X	X	X
Longdisp	Longitudinal dispersivity	X	X	X
Bulkdens	Bulk density	X	X	X
Imporosity	Immobile porosity	X	X	X
Material	Material	X		

Polygon Properties for MODFLOW/MT3D Layer Coverage.

Observation Coverage

When importing shapefiles into an observation coverage, information should exist in the Int (interval) and Conf (confidence) fields, or in the Stdev (standard deviation) field but not both. The confidence value should be an integer between 0 and 100 (i.e., 95 means 95% confidence). GMS will assign default values if there is any information missing.

Type	Name	Z	Layer	Value	Int	Conf	Stdev
Obs	X	X	X	X	X	X	X

Point Properties for Observation Coverage.

Shapefiles

One of the most common methods for creating feature objects is to import a shapefile. The concept of a shapefile was established by Environmental Systems Research Institute (ESRI) [1] in their ArcView® program and it has become the defacto standard for sharing GIS vector data (points, lines, and polygons).

A shape file is actually comprised of three or more files. The primary file is the *.shp and it contains the geometric information (coordinates and if necessary connectivity of the points, lines, polygons). The *.dbf file is a standard database file and stores the attributes of the feature objects. Finally, there will be a *.shx file which is an indexing file. There may be a few other files that accompany the shapefile and so you should always move them around together if you are copying or moving them to a new directory.

Only one "theme" or type of feature can exist in a shapefile. For example you cannot store points and polygons in the shapefile, or boundaries and so you may be required to import multiple files.

XMS software includes all of the tools necessary to import shapefiles and convert the geometric and attribute information into feature objects. This can be done by directly opening the shapefile and converting to feature objects in the active coverage or by loading the shapefile in the GIS module.

Import From Database

The "Database Import Wizard" allows users to access data stored in a database and import it. The wizard is invoked by selecting the ***Import from Database*** command from the *File* menu. The data types that can be imported by the **Database Import Wizard** are the same data types that are supported by the "Text Import Wizard".

The *Database Import Wizard* has four steps:

Step 1 - Connecting to a Database

Step 1 of the wizard lets you set up a connection to a database. To connect to a database either on the same computer or on a network press the **Connect to Database** button. Once a database connection is created, a path to the database and the different tables in the database are displayed. When a table is selected its columns are displayed along with the number of rows in that table.

Step 2 - Querying Information from a Database Table

Step 2 allows you to create, copy, delete, and import queries that retrieve data from a database. To help in writing the queries, the tables in the database are displayed, as well as the columns in the selected table.

Some SQL Basics The query is an SQL (Structured Query Language) statement. The SQL statement is entered in the **Query SQL statement** edit field. An example of an SQL statement would be: “SELECT x, y, z, toluene FROM multipledatasets”. This statement means that columns x, y, z, and toluene from the table multipledatasets will be retrieved from a database. SQL statements are case sensitive. SQL statements also require brackets around table or column names that have spaces. For example, to query data from a table titled “x coordinate” in the SQL statement it would be written as [x coordinate]. A full explanation of SQL is beyond the scope of this document.

You can write a short description for each query in the **Query description** edit field. The **New** button creates a new query that has a default name, description, and SQL statement. The **Copy** button creates a copy of the currently selected query. The **Delete** button deletes the currently selected query.

The queries you define are saved automatically by GMS in a file called gmsquery.ini, located in the folder where GMS is installed. The **Import** button allows you to import a list of queries from any file that follows the same

format as the gmsquery.ini file.

Step 3 - Viewing the Results of the Query

Step 3 displays the results of the database query. Only the first 20 rows are displayed in a spreadsheet. If the results are not what you wanted, you can go back to Step 2 and modify the query.

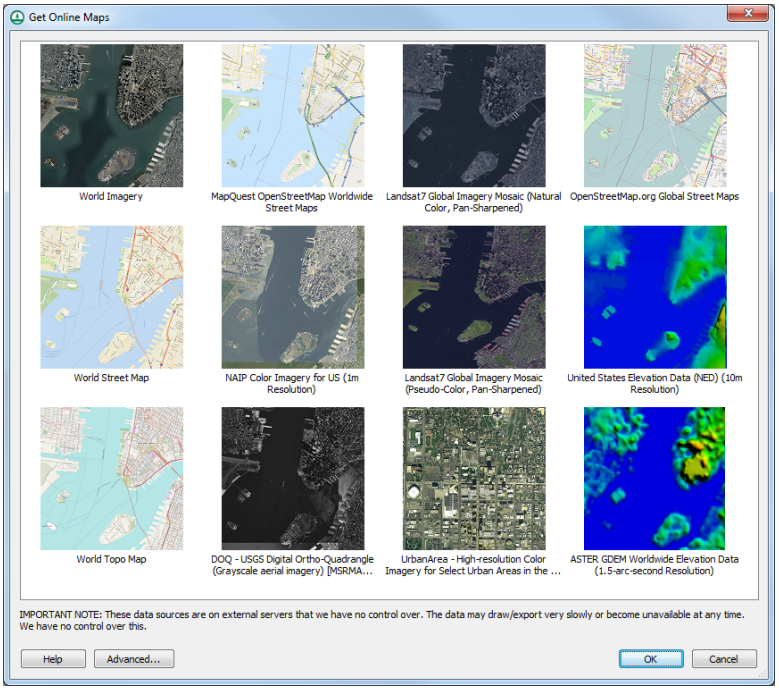
Step 4 - Assigning Column Types

The first 20 lines of the file are displayed in a spreadsheet according to the file outline specified in step 1. This step lets you pick what kind of data you are importing (see Supported File Formats). A "no data flag" can be specified for the file. This is a number that, when encountered in the file, tells GMS to mark the value as "NULL" or "no data".

The data in the columns are identified by selecting the type in the combo box at the top of each column in the spreadsheet. If a row of headings exists, GMS will automatically select the proper type if it recognizes the heading. Otherwise they are Not Mapped by default. The available column types changes depending on the GMS data type selected. Certain column types must be mapped for each file format before the user can progress to the next step in the wizard. The name of each column is changed by editing the Header cell.

After the data have been imported, the GMS:Coordinate Transformation tools can be used to transform and translate the data.

Get Online Maps



Get Online Maps Dialog

The **Get Online Maps** button allows you to select online data from a variety of different sources. Once online maps has been selected, the data resolution will be automatically adjusted based on your zoom parameters. Online maps can only be viewed in plan view.

Online maps are raster datasets that can contain imagery, elevation, or land use information. If you have online maps in your program, you can right-click on each of the maps to convert them to static images that can be saved to your local hard drive. You can convert or interpolate online maps containing elevation data to various elevation formats.

Note that online data sources are on external servers that we have no control over. The data may draw/export very slowly or become unavailable at any time. We have no control over this.

The **Advanced** button allows you to select from other data sources and to use other online data query functions that may not be fully supported. In the *Advanced* dialog, the **Add Sources From File** button allows you to add new Web Map Service (WMS) sources from an external text file.

Exporting Non-native GMS Files

Exporting Non-native GMS Files

GMS can export files that can be used in other software. There are two methods to export data from GMS. Data can be exported through the Save As dialog or by right-clicking on items in the Project Explorer.

Exporting through Save Dialog

The file types that can be exported are shown in the table below. To export a particular file type, the file filter corresponding to that file type should be selected in the *Save* dialog.

File Type	File Ext	Description
Shapefile	*.shp	Creates a shapefile from either the active coverage, or from the active 2D scatter point set, depending on the choice made in a pop-up dialog.
Tabular Text Files	*.txt	Exports the active 2D or 3D scatter point set as a tab-delimited text file.
Arc/Info Grid	*.asc	Exports a 2D grid to an ASCII grid file that can be read by Arc/Info or ArcView. See Importing/Exporting GIS Grids for more information.
GRASS Grid	.ggd	Exports a 2D grid to using the GRASS ASCII grid file format. See Importing/Exporting GIS Grids for more information.

Exporting from the Project Explorer

Item in the Project Explorer can be exported to different formats by right-clicking on an item and selecting the ***Export*** command. The table below lists the file formats that will be shown depending on the type of Project Explorer object selected:

Project Explorer Object	File Ext	Description
TINs	*.tin	Text GMS TIN File
DXF/DWG	*.dxf, .dwg	Saves the current CAD data to a DXF or DWG file. You can convert your GMS data to CAD data by using the ***Data→CAD*** command in the CAD menu in the Map module.
Boreholes	*.bor	Text GMS Borehole File, Text Tab Delimited Borehole File
Solids	*.sol	Text GMS Solid File
2D Mesh Data	*.2dm	Text GMS 2D Mesh File
2D Grid Data	*.2dg, *.asc, *.ggd	Text GMS 2D Grid File, ARC/INFO Ascii Grid Files, Grass Grid Files
2D Scatter Sets	.xy,.txt,.shp	Text GMS 2D Scatter Point File, Text Tab Delimited 2D Scatter Point File, Shapefile
3D Mesh Data	*.3dm	Text GMS 3D Mesh File
3D Grid Data	*.3dg, *.asc, *.ggd	Text GMS 3D Grid File, ARC/INFO Ascii Grid Files, Grass Grid Files
3D Scatter Sets	*.xyz, *.txt, *.shp	Text GMS 3D Scatter Point File, Text Tab Delimited 3D Scatter Point File, Shapefile
Coverages	*.shp	Shapefile
Datasets	*.dat, *.dat	Binary Dataset Files, ASCII Dataset Files

Particle Sets	*.txt	Text Tab Delimited Pathlines

Multiple Objects

If multiple objects are selected and they are all of the same type, they will all be exported to the same text file.

Folders

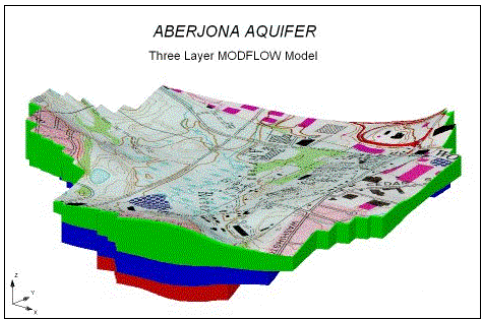
If a folder of objects is selected, then all of the geometric objects in the folder will be exported. Datasets will not be exported. If there are subfolders in the folder, all of the geometric objects will be recursively exported in all subfolders.

Images

An image is typically a scanned map or aerial photo in TIFF or JPEG format. Images can be imported to GMS and displayed in the background to aid in the placement of objects as they are being constructed or simply to enhance a plot. Images can also be draped or "texture mapped" onto a TIN, 2D grid, 2D mesh, or the top of a 3D grid or 3D mesh.

Image display options are changed in the Project Explorer. The options are as follows:

- **Visibility –** The Visibility of an Image is turned off by toggling the check box next to the image in the Project Explorer.
- **Transparency –** The Transparency of the images can be changed by right-clicking on the image folder in the Project Explorer and selecting the ***Transparency*** command.
- **Crop Collar –** USGS quad sheet collars can be removed from an image by right-clicking on an image in the Project Explorer and selecting the ***Crop Collar*** Command.



Scanned USGS Quad Map Texture Mapped to the Top of a 3D Grid.

Importing an Image

Images are read into GMS by using the ***File*** | ***Open*** command. GMS supports many different image file types.

Image Pyramids

Image pyramids can be created when an image is imported into GMS. Image pyramids are multiple resolution copies of the original image that are created and saved to speed up display. With pyramids, a lower resolution copy of the image displays quickly when drawing the entire image. As you zoom in, copies of the original image resampled at finer resolutions are then drawn. This option allows for optimal performance and clear image display.

Multiple Images

GMS allows more than one image at a time; however, you will want to insure that each image is in the same coordinate system.

Geo-Referenced Images

Some images have the coordinate system embedded in the file. Also, some images will have a separate file called a world file (for example: a tiff world file, *.tfw). These files are "geo-referenced." When these files are opened GMS automatically registers the image to the real world coordinate location specified. In the case where a separate world file is used, GMS will automatically find it and register the image if the world file has the same filename prefix as the image file and is in the same folder.

If the image file is not geo-referenced then you will have to register the image manually. (See Registering an Image)

When the GMS project is saved, a link to the image is saved in the project file, along with the current image registration information so that the image is re-registered to the same coordinates every time the project is opened. The original image file and world file (if one exists) are not altered.

Exporting Images

An image can be exported by right-clicking on the image in the *Project Explorer* and selecting the *Export* command. When exporting, the image file type and the resolution can be specified. Only the image which is open in GMS is exported, not the entire *Graphics Window*.

The entire *Graphics Window* can also be saved to an image file via the *File|Save As* command. A number of different image file types are supported such as: *.bmp, *.png, *.jpg, *.tif.

Texture Mapping Images

GMS allows you to drape an image over the top of a TIN, grid, or mesh. However, only the first image listed in the project explorer will be texture mapped onto the TIN, grid, or mesh. You can change the order of the images in the project explorer by selecting the image and dragging it to a new location in the project explorer.

Image Properties

The *Image Properties* dialog can be opened by right-clicking on an image in the *Project Explorer* and selecting the *Properties* command from the pop-up menu. This dialog displays information about the image, such as name, path, type, pixel size etc.

DEMs / Rasters

See Rasters.

Registering an Image

If an image file is not geo-referenced then the user must define the coordinate system of the image. The register dialog allows the user to specify the coordinate system for the image. When an image is opened, if the image is not self-referenced, XMS attempts to find world file with the same name as the image (*.wld or *.jpgw extension). If neither of these is found, the register dialog opens.

What is Image Registration?

Before an image can be displayed, the image must be "registered" or geo-referenced. Registering an image involves identifying points on the image corresponding to locations with known real world (XY) coordinates. Once these points are identified, they are used to scale and translate the image to the proper location when it is drawn with the other objects in the Graphics Window. If an image is not registered properly, any objects which are created using the background image as a guide will have the wrong coordinates.

Register Image Dialog

An image is registered using the *Register Image* dialog. The main feature of the *Register Image* dialog is a large window in which the image is displayed. Two or three points (shown by "+" symbols) are also displayed in the window. These points are used to identify locations with known real world coordinates. The real world coordinates (X,Y) and image coordinates (U,V) of the registration points are listed in edit fields below the image. The points are moved to the desired locations on the image by dragging the points using the tools described below. Once the points are located, the real world coordinates can be entered in the corresponding edit fields. The dialog contains the following options:

- **2 point or 3 point registration –** Two point registration rotates and uniformly scales an image. Three point registration allows for non-uniform scaling to account for some parallax.
- **Import World File –** Used to import a TIFF world file (*.tfw). A TIFF world file has the information needed to set the (X,Y) and (U,V) coordinates in order to place the image in the correct world coordinates.
- **Image name –** Used to associate a name with the file. This name will appear in the project explorer.

Register Image Dialog Tools

The following tools can be used to help position the registration points:

Tool	Tool Name	Description
	Select Point Tool	The Select Point tool is used to select and drag register points to a location on the map for which real coordinates are known so that they can be entered in the corresponding XY edit fields.
	Zoom Tool	In some cases, it is useful to magnify a portion of the image so that a registration point can be placed with more accuracy. The Zoom Tool is used to zoom in a portion of the image.
	Pan Tool	After zooming in on a portion of the image, the Pan Tool is used to pan the image vertically or horizontally.
	Frame Macro	The Frame Macro is used to automatically center the entire image within the drawing window of the dialog after panning and zooming in on a specific location.

Import World File

The **Import World File** button can be used to automatically define the registration data. A world file is a special file associated with a previously registered image that is exported from ArcView® [1] or Arc/Info® [2]. The file contains registration data that can be used to register the image.

Saving/Reading Image Registration Data

When a project file is saved, a link to the image is saved in the project file, along with the current image registration information so that the image is re-registered to the same coordinates every time the project is opened. The original image file and world file (if one exists) are not altered.

Convert Point Coordinate System

The x, y coordinates of each register point must be specified. If the user has the (x,y) coordinates in a different coordinate system than their project, the coordinates will need to be converted.

GMS Point Conversion

The **Convert Point** button in the image registration dialog will allow the user to convert the coordinates.

SMS Point Conversion

The Single Point Conversion command in the *Edit* menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

An alternative approach is to convert the coordinate system after importing by right clicking on the image in the Project Explorer and choosing ***Coordinate Conversion*** from the right click menu.

WMS Point Conversion

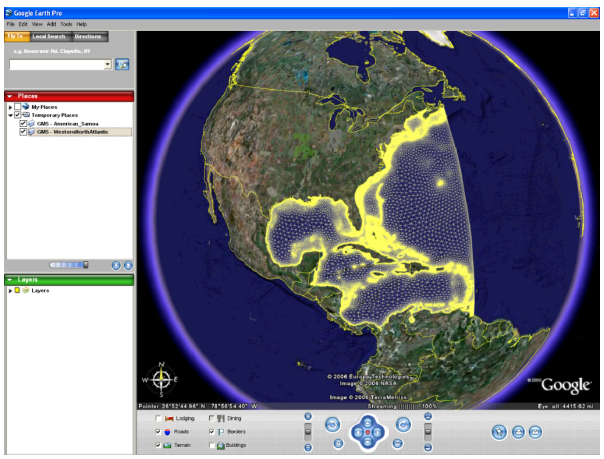
The Single Point Conversion command in the *Edit* menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

References

[1] http://www.esri.com/software/arcview/
[2] http://www.esri.com/software/arcgis/arcinfo/

KMZ files

XMS software can export KMZ files. KMZ [1] files can be imported into Google Earth [2]. KMZ files cannot currently be imported into XMS.



A 2D Mesh inside Google Earth

Raster vs. Vector

The KMZ file format supports both vector data (lines, points, polygons, triangles etc.) and raster data (images). When exporting raster data, the image shown in the XMS main graphics window is saved as a PNG image file with georeferencing data. The data is clipped to match the window bounds of the current view. When exporting vector data, all data displayed, as specified by the display options and project explorer, is exported to a raster KMZ file. The following versions of XMS software support vector export:

- GMS – 7.1
- SMS – 10.1
- WMS – 8.1

How To Export - Screen Capture

- You must be in plan view before you can export a KMZ file.
- You can export a KMZ file by using the standard *File|Save As* dialog and selecting either the **Google Earth© Raster KMZ File (*.kmz)** or **Google Earth© Vector KMZ File (*.kmz)** option in the **Save as type** field.
- To specify a resolution higher than the screen resolution:
  - GMS – Select the **Options** button in the *Save As* dialog.
  - SMS – Change the copy to clipboard scale factor in the Preferences dialog

The background color is made transparent in the exported KMZ file so the Earth can be seen through the overlayed image in Google Earth©.

How To Export – Transient Data Animation

Export a KMZ transient data animation using the Film Loop Wizard. This will export a series of raster images which can be animated in Google Earth©. The background color can be specified in the Film Loop Wizard. It is recommended that the option for no background be used so the Earth can be seen through the overlayed image in Google Earth©. You must be in Plan View to export a KMZ transient data animation.

By default, Coordinated Universal Time (UTC) [3] is assumed when exporting KMZ files. An offset from UTC can be specified. A list of time zone offsets from UTC is given here [4].

See "Viewing a Timeline" on page 90 of the Google Earth© user Guide [5] or here [6] for an explanation of how to change the time zone used by Google Earth©.

Coordinate System

KMZ files contain latitude and longitude information to define the location of the image. If your current coordinate system type is a projection, like UTM for example, and not a geographic system (which uses latitude and longitude), XMS will attempt to determine the latitude and longitude using coordinate conversion. It's possible that the coordinate conversion may fail, or that the distortion from converting from the current coordinate system to a geographic system is too high. In either case, XMS will issue a warning.

Transparency

The background color is made transparent in the exported PNG image which is in the KMZ file. This makes it so that the Earth can be seen through the overlayed image.

See Also

- Official Google Earth website [2]
- KML documentation [1]

Back to XMS

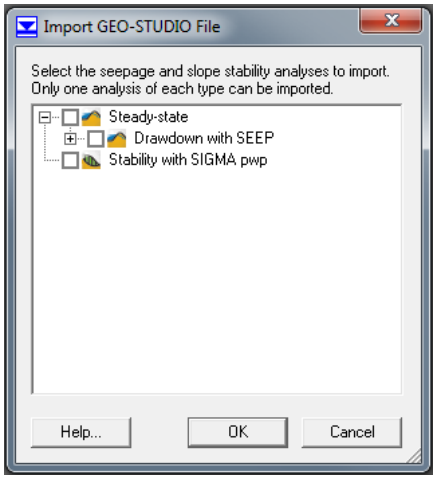
References

[1] http://code.google.com/apis/kml/documentation/
[2] http://earth.google.com/
[3] http://en.wikipedia.org/wiki/Coordinated_Universal_Time
[4] http://en.wikipedia.org/wiki/Timezones
[5] http://earth.google.com/userguide/v4/google_earth_user_guide.pdf
[6] http://earth.google.com/userguide/v4/ug_gps.html

GeoStudio Import

Importing a GeoStudio file

A GeoStudio file may be opened from either the File menu or by drag-dropping the file into the GMS window. A dialog similar to the figure below will appear asking which SEEP or Stability analyses to import. Only one of each method can be selected at once. Click OK when selections have been decided and GMS will display the imported information.



Importing a GeoStudio file

9. Appendix

File Formats

Old GMS file formats

This pdf file [1] documents file formats used by GMS previous to version 6.5. Current versions of GMS can still read these files.

GMS version 6.5 and up

Starting at version 6.5 the GMS project file switched to a binary HDF5 [1] format. The format is proprietary and not documented but you can examine the contents of the project file using a free HDF5 browser [2] application. Some things in the project file, like TINs and Meshes, follow the XMDF standard [3]. Editing the HDF5 project file can corrupt it and make it unreadable by GMS and is highly discouraged.

Import formats

GMS can import various formats as documented on the Text Import Wizard Supported File Formats page.

References

[1] http://www.hdfgroup.org/HDF5/
[2] http://www.hdfgroup.org/hdf-java-html/hdfview/index.html
[3] http://www.xmswiki.com/xms/XMDF

Data Set Files

ASCII Dataset Files

Datasets can be stored to either ASCII or binary files. The default format is binary. Datasets can be saved in ASCII format by right clicking on the dataset in the Project Explorer and selecting the ***Export*** command from the pop up menu. For both file formats, multiple datasets can be stored in a single file and both scalar and vector datasets can be saved to the same file. The file format is identical for 2D and 3D datasets.

For scalar dataset files, one value is listed per vertex, cell, node, or scatter point. The points are listed sequentially in ascending order according to the ids of the nodes, points, vertices, or cells. For vector dataset files, one set of XYZ vector components is listed per vertex, cell, node, or scatter point. If necessary, a set of status flags can be included in the file. If the status flag is false (0), the corresponding item (node, cell, etc.) is inactive. If status flags are not included in the file, it is assumed that all items are active.

The ASCII Dataset file format is as follows:

DATASET // File type identifier

OBJTYPE type // Type of object dataset is associated with

REFTIME reftime // A value corresponding to beginning date/time of dataset

BEGSCL // Beginning of scalar dataset

OBJID id // Object id

ND numdata // Number of data values

NC numcells // Number of cells or elements

NAME "name" // Dataset name

ACTTS time // Marks the active time step

MAPTS time // Marks the time step which is mapped as elevations

TS istat time // Time step of the following data

stat1 // Status flags

stat2

.

.

statnumcells

val1 // Scalar data values

val2

.

.

valnumdata

// Repeat TS card for each time step

ENDDS // End of dataset

BEGVEC // Beginning of vector dataset

VECTYPE type // Vector at node/gridnode or element/cell

OBJID id // Object id

ND numdata // Number of data values

NC numcells // Number of cells or elements

NAME "name" // Data set name

TS istat time // Time step of the following data

stat1 // Status flags

stat2

.

.

statnumcells

vx1 vy1 vz1

vx2 vy2 vz2

.

.

vnumdata vnumdata vnumdata

// Repeat TS card for each time step

ENDDS // End of dataset

// Repeat BEGSCL and BEGVEC sequences for each dataset

Sample ASCII Dataset File:

DATASET

OBJTYPE grid2d

REFTIME 945.348729

BEGSCL

ACTTS 1.00000000e+00

ND 8

NC 8

NAME "trichloroethylene"

TS 1 1.00000000e+00

0

0

0

1

1

1

1

0

0.00000000e+00

0.00000000e+00

0.00000000e+00

3.24000000e+00

4.39000000e+00

2.96000000e+00

7.48000000e+00

0.00000000e+00

ENDDS

BEGVEC

VECTYPE 0

ND 8

NC 8

NAME "velocity"

TS 1 5.00000000e+00

0

0

0

1

1

1

1

0

1.60000000e+01 1.60000000e+01 3.20000000e+01

6.40000000e+01 6.40000000e+01 1.28000000e+02

1.44000000e+02 1.44000000e+02 2.88000000e+02

1.96000000e+02 1.96000000e+02 3.92000000e+02

2.25000000e+02 2.25000000e+02 4.50000000e+02

9.21600000e+03 9.21600000e+03 1.84320000e+04

9.60400000e+03 9.60400000e+03 1.92080000e+04

9.80100000e+03 9.80100000e+03 1.96020000e+04

ENDDS

If variograms have been defined for a dataset or time step of a dataset within GMS, the variograms are saved in the dataset file. The variogram cards are not documented.

The card types used in the scalar data set file format are as follows:

Card Type	**DATASET**
Decription	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**OBJTYPE**
Decription	Identifies the type of objects that the datasets in the file are associated with.
Required	YES. If card does not exist, the file can only be read through the Data Browser. The datasets would then be assigned to the objects corresponding to the active module.
Format	OBJTYPE type
Sample	OBJTYPE tin

Field	*Variable*	*Value*	*Description*
1	type	tin	TINs
		mesh2d	2D meshes
		grid2d	2D grids
		scat2d	2D scatter points
		mesh3d	3D meshes
		grid3d	3D grids
		scat3d	3D scatter points

Card Type	**REFTIME**
Decription	A value corresponding to the beginning date/time of the dataset.
Required	NO
Format	REFTIME reftime
Sample	REFTIME 3982.897459

Field	*Variable*	*Value*	*Description*
1	reftime	" +/- "	A value corresponding to the beginning date/time of the dataset.

Card Type	**BEGSCL**
Decription	Scalar dataset file identifier. Marks beginning of scalar dataset. No fields.
Required	YES

Card Type	**BEGVEC**
Decription	Vector dataset file identifier. Marks beginning of vector dataset. No fields.
Required	YES

Card Type	**ACTTS**
Decription	Used to mark the active dataset. The card should be placed after the BEGSCL or BEGVEC card of the active dataset and the active time step should be listed.
Required	NO
Format	ACTTS time
Sample	ACTTS 0.00

Field	*Variable*	*Value*	*Description*
1	time	" +/- "	The time corresponding to the active time step. Use 0.0 for steady state datasets.

Card Type	**MAPTS**
Decription	Used to mark the dataset which is mapped to the object elevations. The card should be placed after the BEGSCL or BEGVEC card of the mapped dataset and the mapped time step should be listed.
Required	NO
Format	MAPTS time
Sample	MAPTS 0.00

Field	*Variable*	*Value*	*Description*
1	time	" +/- "	The time corresponding to the mapped time step. Use 0.0 for steady state datasets.

Card Type	**OBJID**
Decription	The unique id of the object the dataset is associated with.
Required	This card is required for datasets associated with TINs and scatter point sets.
Format	OBJID id
Sample	OBJID 2383

Field	*Variable*	*Value*	*Description*
1	id	" + "	The unique id of the object.

Card Type	**VECTYPE**
Decription	Identifies the type of vector data that will be read and where to apply it.
Required	This card is only required if the vector data is associated with elements/cells. If this card is not present, it is assumed that the data are associated with nodes/gridnodes.
Format	VECTYPE tpye
Sample	VECTYPE 0

Field	*Variable*	*Value*	*Description*
1	type	" 0,1 "	0 = The vectors will be applied to the nodes/gridnodes. 1 = The vectors will be applied to the elements/cells

Card Type	**ND**
Decription	The number of data values that will be listed per time step. This number should correspond to the total number of vertices, nodes, cells centers (cell-centered grid), cell corners (mesh-centered grid), maximum node id (meshes) or scatter points.
Required	YES
Format	ND numdata
Sample	ND 10098

Field	*Variable*	*Value*	*Description*
1	numdata	" + "	The number of items. At each time step, numdata values are printed.

Card Type	**NC**
Decription	This number should correspond to the maximum element id (meshes) or the number of cells (grids).
Required	YES
Format	NC numcells
Sample	NC 3982

Field	*Variable*	*Value*	*Description*
1	numcells	" + "	The number of elements or cells

Card Type	**NAME**
Decription	The name of the dataset
Required	YES
Format	NAME "name"
Sample	NAME "Total head"

Field	*Variable*	*Value*	*Description*
1	"name"	str	The name of the dataset in double quotes.

Card Type	**TS**
Decription	Marks the beginning of a new time step, indicates if stat flags are given, and defines the time step value, status flags, and scalar data values for each item.
Required	YES
Format	TS istat time stat1 stat2 . . stat numcells val1 val2 . . valnumdata
Sample	TS 1 12.5 0 1 1 1 34.5 74.3 58.4 72.9

Field	*Variable*	*Value*	*Description*
1	istat	0,1	-
2	time	+	The time step value. If only one time step exists, the time is not required
3 - (nd+2)	stat	0,1	The status of each item. If active, stat=1. If inactive stat=0. Omitted if i=0 on STAT card.
(nd+2) - (2nd+2)	val	" +/- "	The scalar data values of each item

Cardtype	*ENDDS*
Description	Marks the end of a scalar opr vector dataset. No fields
Required	YES

Binary Dataset Files

Datasets saved with a GMS project are saved in the binary format. The binary format is patterned after the ASCII format in that the data are grouped into "cards". However, the cards are identified by a number rather than a card title. The card ids are four byte integers. The binary format is as follows:

Card	Item	Size	Description
	version	4 byte integer	The GMS binary dataset file format version. value = 3000.

100	object type	4 byte integer	Identifies the type of objects that the datasets in the file are associated with. Options are as follows:
			`1 TINs`
			`2 Boreholes`
			`3 2D meshes`
			`4 2D grids`
			`5 2D scatter points`
			`6 3D meshes`
			`7 3D grids`
			`8 3D scatter points`

110	SFLT	4 byte integer	The number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).
120	SFLG	4 byte integer	The number of bytes that will be used in the remainder of the file for status flags.
130 or 140	BEGSCL or BEGVEC		Marks the beginning of a set of cards defining a scalar or vector dataset.
220	ACTTS	SFLT real	Marks the active dataset and time step
230	MAPTS	SFLT real	Marks the mapped dataset and time step
150	VECTYPE	4 byte integer	(0 or 1) In the case of vector dataset files, indicates whether the vectors will be applied at the nodes/gridnodes or the elements/cells.
160	OBJID	4 byte integer	The id of the associated object. Value is ignored for grids and meshes.
170	NUMDATA	4 byte integer	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered grid) or scatter points.
180	NUMCELLS	4 byte integer	This number should correspond to the number of elements (meshes) or the number of cells (mesh-centered grids). Value is ignored for other object types.

190	NAME	40 bytes	The name of the dataset. Use one character per byte. Mark the end of the string with the '\0' character.
195	REFTIME	8 bytes	A value corresponding to the beginning date/time of the dataset.
200	TS		Marks the beginning of a time step.
	ISTAT	SFLG integer	(0 or 1) Indicates whether or not status flags will be included in the file.
	TIME	SFLT real	Time corresponding to the time step.
	statflag1	SFLG integer	Status flag (0 or 1) for node 1
	statflag2	SFLG integer	Status flag (0 or 1) for node 2
...			
	val1	SFLT real	Scalar value for item 1
	val2	SFLT real	Scalar value for item 2
...			
			Repeat card 200 for each timestep in the dataset.
210	ENDDS		Signal the end of a set of cards defining a dataset.

The cards in the binary dataset file are as follows:

Card Type	**Version**
Card Id	3000
Description	File type identifier. No fields.
Required	Yes

Card Type	**ObjType**
Card Id	100
Description	Identifies the type of objects that the datasets in the file are associated with.
Required	YES. If card does not exist, the file can only be read through the Data Browser. The datasets would then be assigned to the objects corresponding to the active module.

Feild	*Variable*	*Size*	*Value*	*Description*
1	id	4 byte int	1,2,3,4,5,6,7,8,	TINs, Boreholes, 2D meshes, 2D grids, 2D scatter points, 3D meshes, 3D grids, 3D scatter points

Using Vector and Scalar Data with Grids

For meshes, TINs, and scatter point sets, the order that the values are listed in the file is simply the sequential order of the node, TIN, or scatter point ids. However, vector and scalar data can also be associated with the nodes or cells of a 2D or 3D grid. T For 2D grids, data values are ordered using a row-column (I-J) priority. For 3D grids, data values are ordered using a layer-row-column (K-I-J) priority.

The following C source code examples illustrate how a 2D or 3D array of scalar values corresponding to the nodes of a grid would be written to the main portion of an ASCII scalar file.

2D Grid Example:

```
for(i=0; i<nrow; i++){

for(j=0; j<ncol; j++){
fprintf(fp, "%f\n", scalar[i][j]);

}

}
```

3D Grid Example:

```
for(k=0; k<nlay; k++){

for(i=0; i<nrow; i++){

for(j=0; j<ncol; j++){
fprintf(fp, "%f\n", scalar[k][i][j]);

}

}

}
```

Bugfixes GMS

Important Note

If you download and install the latest update and your software maintenance has expired, you will not be able to run that install of GMS. You will need to install an earlier build of GMS or purchase maintenance. Please make sure your GMS maintenance license has not expired before downloading. You can determine your maintenance expiration date by selecting the Help | Register command from GMS. You can renew your maintenance by visiting Aquaveo's web page [1] or by contacting a sales representative. Prices and other information about renewing your license are available on the GMS Pricing Page [2]. The latest update to GMS can be downloaded here [3].

Intermediate Release GMS 9.0.5 - April 17, 2013

Bug ID-Date Submitted-Updated-Summary

- 6082 2013-03-25 2013-04-09 Reaction Parameter Values in RT3D Chemical Reaction Package get Mixed up when mapped to cells User: Terrence Stanley
- 6030 2013-03-12 2013-03-12 SEEP2D Flow Lines Not Updated
- 6096 2013-04-02 2013-04-02 GMS 2Mf2K Exports Bad Array Values when Multiplier is Zero
- 6042 2013-03-13 2013-04-08 GMS Model Checker Suppresses Errors When you Check the Option to Suppress Warnings
- 6083 2013-03-26 2013-04-09 Not Enough Memory Message when Opening GMS Super File User: Rob Healy User: Rob Healy
- 6066 2013-03-21 2013-04-09 VK/VANI does not Display Final Array Values from Pilot Points
- 6118 2013-04-09 2013-04-11 GMS Crash when Saving Project
- 6137 2013-04-16 2013-04-17 Problems Loading Online Images
- 6136 2013-04-15 2013-04-18 GMS Crashes when you try converting Shapefile to 2D Scatter Points

Intermediate Release GMS 9.0.4 - March 25, 2013

Bug ID-Date Submitted-Updated-Summary

- 5885 2-1-13 2-5-13 Pathline to Arcs does not create Arcs
- 5877 1-31-13 2-4-13 Option to generate MODPATH particles at selected wells permanently grayed out
- 5829 1-15-13 2-1-13 velocity vectors not displaying in MODFLOW solution
- 5845 1-22-13 1-31-13 Horizons and Raster's not matching Tutorial
- 5840 1-8-13 1-23-13 crash when interpolating raster's to MODFLOW layers
- 5967 2-26-13 2-27-13 Crash when copying and pasting borehole data into borehole editor
- 5931 2-14-13 2-27-13 GMS Crash when creating new T-Progs Simulation

Intermediate Release GMS 9.0.3 - January 16, 2013

Bug ID-Date Submitted-Updated-Summary

- 5761 2012-12-13 2012-12-17 3D mesh shell doesn't display
- 5739 2012-12-04 2012-12-06 Water table displays as spiky near dry cells in MODFLOW model
- 5758 2012-12-13 2012-12-13 Attempting to drag tin vertex without associated triangle crashes
- 5729 2012-11-30 2013-01-02 Retriangulate after deleting with 2D mesh
- 5730 2012-11-30 2013-01-02 swap edges tool
- 5765 2012-12-14 2013-01-02 Crash when you open and close a MODFLOW array and use the Add K boundary tool
- 5797 2013-01-04-2013-01-07 Crash in Select Data Set Dialog User: William Frederick
- 5759 2012-12-13 2013-01-07 Cannot tell where the Particle Sets folder is exported in GMS
- 5753 2012-12-11 2013-01-02 Can't assign a negative flowrate value for SEEP2D User: Kip Solomon
- 5790 2013-01-02 2013-01-07 Can't get rid of error message when cutting cross sections
- 5770 2012-12-19 2013-01-03 Changing layer to confined and reopening file results in message saying "Error reading flow package file." Iser" Julian Weir

Intermediate Release GMS 9.0.2 - November 28, 2012

Bug ID-Date Submitted-Updated-Summary

- 5702 2012-11-20 2012-11-29 Reading in old map file doesn't set conceptual model type to MODFLOW
- 4141 2011-09-16 2012-11-30 line object: cancel button, anchoring
- 5543 2012-10-04 2012-11-06 GMS repeats the unit codes for the text files in MODFLOW 2005 which causes a crash - User: Julian Weir
- 5580 2012-10-12 2012-11-06 GMS crashes when you hit the Import button in the Define Parameters dialog - User: Christian Johnson
- 5617 2012-10-24 2012-11-07 Convert boreholes to cad
- 5582 2012-10-12 2012-11-07 Active and Qdes times/values different in MODFLOW source sink dialog for same well - User: Julian Weir
- 5633 2012-10-26 2012-11-07 Number of decimal places that user specifies always defaults to 3 when exporting grid as shapefile
- 5681 2012-11-12 2012-11-15 Zone codes revert back to default value of 1 when performing Map -> MODFLOW/MODPATH - User: Christoph Mobus
- 5676 2012-11-09 2012-11-15 Can't move nodes in 3D grids - User: Christoph Mobus
- 5636 2012-10-29 2012-11-15 Interpolate from Transient data to TIN

Intermediate Release GMS 8.3.7 - November 12, 2012

Bug ID-Date Submitted-Updated-Summary

- 5543 2012-10-04 2012-11-06 GMS repeats the unit codes for the text files in MODFLOW 2005 which causes a crash - User: Julian Weir
- 5580 2012-10-12 2012-11-06 GMS crashes when you hit the Import button in the Define Parameters dialog - User: Christian Johnson
- 5617 2012-10-24 2012-11-07 Convert boreholes to cad
- 5597 2012-10-17 2012-11-07 Observed values are plotted multiple times even if GMS is only plotting one solution - User: Chris Mobus
- 5582 2012-10-12 2012-11-07 Active and Qdes times/values different in MODFLOW source sink dialog for same well - User: Julian Weir

- 5632 2012-10-26 2012-11-07 Transient Head observation times all become zero if check and uncheck the date/time option. - User: Colin Smith
- 5633 2012-10-26 2012-11-07 Number of decimal places that user specifies always defaults to 3 when exporting grid as shapefile
- 5681 2012-11-12 2012-11-15 Zone codes revert back to default value of 1 when performing Map -> MODFLOW/MODPATH - User: Christoph Mobus
- 5676 2012-11-09 2012-11-15 Can't move nodes in 3D grids - User: Christoph Mobus

Intermediate Release GMS 9.0.1 – October 18, 2012

Bug ID-Date Submitted-Updated-Summary

- 5365 2012-08-06 2012-08-30 Crash when adding boundary - User: Shuangshuang Xie
- 5509 2012-09-24 2012-09-26 Horizons -> Solids Crash - User: Julie Butler
- 5500 2012-09-20 2012-09-26 GMS crash with the Create Grid Dialog
- 5502 2012-09-20 2012-09-26 2D Interpolation issue
- 5480 2012-09-11 2012-09-26 GMS crashes when attempting to create a 2D xz grid in 3D grid module - User: Carina Carlsen
- 5481 2012-09-11 2012-09-27 GMS always maps a zero pumping rate when you check the well screen option
- 5504 2012-09-21 2012-09-27 MT3D Source/Sink mixing package dialog always shows the type as "river" even if the use stream package option is checked - User: Julian Weir
- 5510 2012-09-24 2012-09-27 GMS averages concentrations between two coverages when an entire conceptual model is mapped to MT3DMS - User: Julian Weir
- 5499 2012-09-20 2012-09-28 Map->MODFLOW well screens, HK by material with HK parameters
- 5451 2012-09-04 2012-09-28 GMS displays weird Cell when Cell is activated and then deactivated - User: Lance Jones
- 5485 2012-09-12 2012-09-28 Velocity vector calculation does not use the porosity from Material IDs
- 5514 2012-09-25 2012-09-28 Contour populate values
- 2702 2010-10-22 2012-09-28 Add RLAMDA1 and RLAMFAC to PEST interface
- 5501 2012-09-20 2012-10-04 parameter estimation option is dimmed out when switching to BCF package - User: Colin Smith
- 5473 2012-09-07 2012-10-04 Text background does not update in display options dialog
- 5475 2012-09-10 2012-10-04 Select intersecting object does not work with multiple polygons
- 5488 2012-09-13 2012-10-04 Annotation tools do not display
- 5531 2012-10-01 2012-10-04 GMS crashes when moving a vertex - User: Christoph Mobus
- 5532 2012-10-02 2012-10-04 Crash when importing .par file that doesn't match model - User: Colin Smith
- 5475 2012-09-10 2012-10-04 Select intersecting object does not work with multiple polygons
- 5533 2012-10-02 2012-10-04 Interpolating scatter to MODFLOW layers is not working
- 5537 2012-10-03 2012-10-09 Crash when closing pump capacity curve window without any data

Intermediate Release GMS 8.3.6 - October 18, 2012

Bug ID-Date Submitted-Updated-Summary

- 5365 2012-08-06 2012-08-30 Crash when adding boundary - User: Shuangshuang Xie
- 5509 2012-09-24 2012-09-26 Horizons -> Solids Crash - User: Julie Butler
- 5500 2012-09-20 2012-09-26 GMS crash with the Create Grid Dialog
- 5502 2012-09-20 2012-09-26 2D Interpolation issue
- 5480 2012-09-11 2012-09-26 GMS crashes when attempting to create a 2D xz grid in 3D grid module - User: Carina Carlsen
- 5481 2012-09-11 2012-09-27 GMS always maps a zero pumping rate when you check the well screen option
- 5504 2012-09-21 2012-09-27 MT3D Source/Sink mixing package dialog always shows the type as "river" even if the use stream package option is checked - User: Julian Weir
- 5510 2012-09-24 2012-09-27 GMS averages concentrations between two coverages when an entire conceptual model is mapped to MT3DMS - User: Julian Weir
- 5499 2012-09-20 2012-09-28 Map->MODFLOW well screens, HK by material with HK parameters
- 5451 2012-09-04 2012-09-28 GMS displays weird Cell when Cell is activated and then deactivated - User: Lance Jones
- 5485 2012-09-12 2012-09-28 Velocity vector calculation does not use the porosity from Material IDs
- 5514 2012-09-25 2012-09-28 Contour populate values
- 5501 2012-09-20 2012-10-04 parameter estimation option is dimmed out when switching to BCF package - User: Colin Smith
- 5473 2012-09-07 2012-10-04 Text background does not update in display options dialog
- 5475 2012-09-10 2012-10-04 Select intersecting object does not work with multiple polygons
- 5488 2012-09-13 2012-10-04 Annotation tools do not display
- 5531 2012-10-01 2012-10-04 GMS crashes when moving a vertex - User: Christoph Mobus
- 5532 2012-10-02 2012-10-04 Crash when importing .par file that doesn't match model - User: Colin Smith
- 5475 2012-09-10 2012-10-04 Select intersecting object does not work with multiple polygons
- 5537 2012-10-03 2012-10-09 Crash when closing pump capacity curve window without any data

Intermediate Release GMS 8.3.5 - Sept 21, 2012

Bug ID-Date Submitted-Updated-Summary

- 5361 2012-08-03 2012-08-30 Intensity color ramp not saved
- 5366 2012-08-06 2012-08-30 Crash when redistributing grid cells in GMS - User: Shuangshuang Xie
- 5428 2012-08-28 2012-09-04 Source/Sinks toggles are mixed up when creating Flow Budget versus Time plots in GMS - User: Christopher Zell
- 5351 2012-08-01 2012-09-04 Projection changed when importing an image
- 5403 2012-08-16 2012-09-05 Warning window does not have enough info
- 5363 2012-08-03 2012-09-06 GMS not reading the SFR2 input data correctly - User: Mary Halstead
- 5401 2012-08-16 2012-09-06 values less than 1e-6 not displayed in Edit XYZF toolbar - User: Christoph Mobus
- 5404 2012-08-16 2012-09-06 color for the borehole edges cannot be changed - User: Tatevik Tadevosyan
- 5320 2012-07-23 2012-09-06 MNW2 wells don't show up in Sources/Sinks dialog
- 5458 2012-09-04 2012-09-10 Flow budget plot bugs - Nightly Build On August 30th
- 5459 2012-09-04 2012-09-10 MNW2 Output flow budget changed
- 5427 2012-08-28 2012-09-11 Open in current project option does not work
- 5377 2012-08-08 2012-09-11 spreadsheet values don't update - User: kevin couch
- 5365 2012-08-06 2012-08-30 Crash when adding boundary - User: Shuangshuang Xie

Intermediate Release GMS 8.3.4 - Aug 24, 2012

Bug ID-Date Submitted-Updated-Summary

- 4714 2012-02-14 2012-08-14 Vertical conductivity/vertical anisotropy units not updating - User: Shuangshuang Xie
- 5282 2012-07-16 2012-08-14 CCF file reports as invalid when SUB package VerticalDisplacement data set included
- 5316 2012-07-23 2012-08-14 Stochastic Options crash
- 5256 2012-07-02 2012-08-14 mf2k5_h5_parallel.exe does not work
- 5261 2012-07-02 2012-08-14 MODFLOW does not run in Stoch Params tutorial
- 5314 2012-07-23 2012-08-14 Units label for LAK leakance in map
- 5089 2012-05-08 2012-08-14 GMS stalls while using Online Maps
- 5300 2012-07-18 2012-08-14 Water table symbol is a 'D' - User: Beth Rowan
- 5307 2012-07-20 2012-08-14 Reseting factory preferences in 32 bit change model execuable paths in 64 bit
- 5203 2012-06-18 2012-08-14 File reference for model executables is wrong
- 5347 2012-07-31 2012-08-14 file says H5 file corrupt or invalid when trying to open
- 5301 2012-07-18 2012-08-14 Borehole material transparency does not work (always solid) - User: Beth Rowan
- 5357 2012-08-02 2012-08-14 GMS crashes when deleting conceptual model - User: Christoph Moebus
- 5349 2012-07-31 2012-08-15 crash when vertical displacement times differs from head or drawdown
- 5194 2012-06-13 2012-08-15 Error when running femwater
- 5359 2012-08-02 2012-08-16 crash when mapping to MT3D - User: James Wieck
- 5387 2012-08-13 2012-08-16 MODFLOW files that run in USGS MODFLOW do not load into GMS - User: Christoph Moebus
- 5292 2012-07-17 2012-08-17 Model checker doesn't pay attention to use last checking for zero values
- 5341 2012-07-30 2012-08-22 MODPATH fails with multiple steady state stress periods

Intermediate Release GMS 8.3.3 - July 19, 2012

Bug ID-Date Submitted-Updated-Summary

- 5210 2012-06-19 2012-06-25 reset factory preferences fails on model executables page
- 4792 2012-03-02 2012-07-11 MT3D inefficient io with SEAWAT - User: David Li
- 5222 2012-06-22 012-07-11 translator error with LMG solver
- 5231 2012-06-27 2012-07-11 Delete vertex crash when retriangulate on delete is on
- 5241 2012-07-02 2012-07-11 dry cell flag being used to calculate root mean squared value when using MF2K5 - User: Tran Vu Long
- 5192 2012-06-12 2012-07-11 Mesh centered 3D grid crashes
- 5211 2012-06-19 2012-07-11 Extra Legend Appearing
- 4673 2012-02-03 2012-07-11 "Unable to contact server" error message when trying to download SRTM data from Web Services - User: John Quinn
- 5102 2012-05-11 2012-07-11 Importing from an Excel Database - User: Fady Hadba
- 5267 2012-07-09 2012-07-17 Cannot open the CCF results
- 5270 2012-07-11 2012-07-17 No gage files appear
- 5267 2012-07-09 2012-07-17 Cannot open the CCF results
- 5273 2012-07-12 2012-07-17 premature end of ccf2 file
- 5283 2012-07-16 2012-07-17 Cannot open STR properties
- 5284 2012-07-16 2012-07-17 Restore factory preferences not working correctly
- 5281 2012-07-16 2012-07-17 Invalid CCF file using double precision
- 4692 2012-02-09 2012-07-17 View values not correct and other issues

- 5290 2012-07-16 2012-07-17 CCF -> Data sets result in data sets with bad max and min values

Intermediate Release GMS 8.2.6 - July 18, 2012

Bug ID-Date Submitted-Updated-Summary

- 5222 2012-06-22 012-07-11 translator error with LMG solver
- 5231 2012-06-27 2012-07-11 Delete vertex crash when retriangulate on delete is on
- 5241 2012-07-02 2012-07-11 dry cell flag being used to calculate root mean squared value when using MF2K5 - User: Tran Vu Long
- 5192 2012-06-12 2012-07-11 Mesh centered 3D grid crashes
- 5211 2012-06-19 2012-07-11 Extra Legend Appearing
- 5102 2012-05-11 2012-07-11 Importing from an Excel Database - User: Fady Hadba

Intermediate Release GMS 8.3.2 - June 21, 2012

Bug ID-Date Submitted-Updated-Summary

- 4925 2012-03-30 2012-05-03 Incorrect LAK package symbol elevations
- 4790 2012-03-01 2012-05-03 VSC package dialog in SEAWAT
- 4848 2012-03-14 2012-05-03 Run ZONEBUDGET tool does not run on some computers - User: Freddy Cortez
- 4941 2012-04-03 2012-05-03 GMS crashes when attempting to enter information the number of instances in the MODFLOW - Parameters - User: Caroline Ballard
- 4948 2012-04-04 2012-05-03 crash when attempting to enter a new stochastic parameter in MODFLOW | Parameters window - User: Caroline Ballard
- 4951 2012-04-04 2012-05-03 GMS freezes (or takes several hours) while trying to save a project - User: Julian Weir
- 5051 2012-05-01 2012-05-03 running mf2k without model wrapper gives error message
- 5049 2012-05-01 2012-05-03 Geostatistics 32 animation does not work
- 4958 2012-04-06 2012-05-03 model checker crashes after deleting model
- 4965 2012-04-10 2012-05-03 Observation files print to SEAWAT Folder when running SEAWAT - User: Medhat El Bihery
- 5042 2012-04-26 2012-05-03 dialog resulting from opening VK/VANI array from project explorer is labeled wrong when using “Specify anisotropy factors”
- 5127 2012-05-21 2012-06-21 horizontal MNW2 wells can cause MODFLOW to exit with an allocation error
- 5129 2012-05-22 2012-06-21 pathlines can only be loaded with File | Open (not dragging and dropping does not work) - User: Jim Teo
- 5108 2012-05-16 2012-06-21 “Run MODPATH automatically for steady state models” option controls format of .sum MODPATH output file - User: Caitlin Current
- 5094 2012-05-09 2012-06-21 freeze when trying to build polygons in GMS 8.3.1 - User: Bruce Campbell
- 5087 2012-05-08 2012-06-22 Saving a MODFLOW simulation as a namefile does not create a namefile - User: Mark Bentley
- 5079 2012-05-07 2012-06-22 GMS freezes while attempting to convert shapefile imported through GIS module to 2D scatterpoints - User: Dale Bridgeford
- 5083 2012-05-07 2012-06-22 Change in font type, style, or size, in display options for nodes does not actually change the font - User: Carina Carlsen
- 5058 2012-05-02 2012-06-22 Error when saving an animation
- 5125 2012-05-21 2012-06-22 Can't delete entry for boreline (MNW2) in the conceptual model
- 5027 2012-04-24 2012-06-22 TIN legend disappears when zooming

- 4934 2012-04-02 2012-06-22 MT3D timer always says "1 seconds remaining total" even when it still requires many minutes to run - User: Bernard Brixel
- 5139 2012-05-29 2012-06-22 well screen looks differently when hiding other coverages
- 5142 2012-05-30 2012-06-22 MODFLOW model with parameter clusters fails to import
- 5126 2012-05-21 2012-06-22 GMS does not map MNW2 horizontal well screen (same elevation with layer's border)
- 5181 2012-06-08 2012-06-22 "attempted an unsupported operation" error when selecting "Scatter Points | Scatter Point Settings" with certain modules - User: Michael Wallace
- 5144 2012-05-31 2012-06-25 GMS returns an error saying that it cannot open the namefile then crashes - User: Bruce Campbell
- 5210 2012-06-19 2012-06-25 reset factory preferences fails on model executables page

Intermediate Release GMS 8.2.5 - June 20, 2012

Bug ID-Date Submitted-Updated-Summary

- 4790 2012-03-01 2012-05-03 VSC package dialog in SEAWAT
- 4848 2012-03-14 2012-05-03 Run ZONEBUDGET tool does not run on some computers - User: Freddy Cortez
- 4941 2012-04-03 2012-05-03 GMS crashes when attempting to enter information the number of instances in the MODFLOW - Parameters - User: Caroline Ballard
- 4948 2012-04-04 2012-05-03 crash when attempting to enter a new stochastic parameter in MODFLOW | Parameters window - User: Caroline Ballard
- 5051 2012-05-01 2012-05-03 running mf2k without model wrapper gives error message
- 4965 2012-04-10 2012-05-03 Observation files print to SEAWAT Folder when running SEAWAT - User: Medhat El Bihery
- 5129 2012-05-22 2012-06-21 pathlines can only be loaded with File | Open (not dragging and dropping does not work) - User: Jim Teo
- 5108 2012-05-16 2012-06-21 “Run MODPATH automatically for steady state models” option controls format of .sum MODPATH output file - User: Caitlin Current
- 5087 2012-05-08 2012-06-22 Saving a MODFLOW simulation as a namefile does not create a namefile - User: Mark Bentley
- 5079 2012-05-07 2012-06-22 GMS freezes while attempting to convert shapefile imported through GIS module to 2D scatterpoints - User: Dale Bridgeford
- 5027 2012-04-24 2012-06-22 TIN legend disappears when zooming
- 5181 2012-06-08 2012-06-22 "attempted an unsupported operation" error when selecting "Scatter Points | Scatter Point Settings" with certain modules - User: Michael Wallace
- 5144 2012-05-31 2012-06-25 GMS returns an error saying that it cannot open the namefile then crashes - User: Bruce Campbell

Intermediate Release GMS 8.3.1 – April 26, 2012

- 3D scatter does not interpolate to grid correctly
- crash when saving the project after changing the subset value - User: Julian Weir
- north arrow path cannot change in "edit | preferences" window - User: Mark Bentley
- ZONEBUDGET last line of CSV2 file not displayed
- changing pilot point value and rerunning MODFLOW produces same results as before the change - User:Julian Weir
- "error opening data set" after adding additional pilot point and rerunning MODFLOW - User: Julian Weir
- Pilot point starting values not saved accurately - User:Tony Melcher
- rotate tool jumps far away with slight change while certain cross sections are displayed - User: Daniel Fisher
- dry cells symbol shown in non-dry cells when selecting drawdown data set - User: Saeed Jorat
- Crash when importing files created in GMS 3.0 - User: Gary Gregory

Intermediate Release GMS 8.2.4 - April 26, 2012

- 3D scatter does not interpolate to grid correctly
- crash when saving the project after changing the subset value - User: Julian Weir
- north arrow path cannot change in "edit | preferences" window - User: Mark Bentley
- ZONEBUDGET last line of CSV2 file not displayed
- changing pilot point value and rerunning MODFLOW produces same results as before the change - User:Julian Weir
- "error opening data set" after adding additional pilot point and rerunning MODFLOW - User: Julian Weir
- Pilot point starting values not saved accurately - User:Tony Melcher
- rotate tool jumps far away with slight change while certain cross sections are displayed - User: Daniel Fisher
- dry cells symbol shown in non-dry cells when selecting drawdown data set - User: Saeed Jorat
- Crash when importing files created in GMS 3.0 - User: Gary Gregory

Intermediate Release GMS 8.2.3 - March 22, 2012

- Data set calculator and scientific notation
- Bad Help button links
- Image does not load correctly
- Animations and dry cells - User: John Czarnecki
- observation point values not printed correctly to MT3D Transient Observation file (*.tob) - User: Greg Pohll
- Duplicating display themes crashes GMS
- annotation line properties help button crashes visual studio
- framing on a 3D shapefile doesn't work
- Coverage Setup Dialog Modified when opening old GMS Project Files - User: Soheil Afshari

Intermediate Release GMS 8.2.2 - Feb 21, 2012

- FEMWATER doesn't read in the correct card id in 3bc file - User: Julian Weir
- Flow vectors not accurate

Intermediate Release GMS 8.2.1 – Feb 21, 2012

- GMS freeze when mapping to MODFLOW - User: Sarah Lyle
- Grid cell not displaying correctly
- Adding contacts to TIN command causes freeze - User Michael Popp
- FEMWATER - GMS can't read in the solution after FEMWATER run
- FEMWATER - Contour color isn't displayed correctly in GMS
- Scatter set appearing behind grid
- “Boreholes | Cross Section Editor” command always greyed out - User: Colin Smith
- MODFLOW-NWT Tutorial Testing
- Plot Issues with large datasets - User:Chaoying Jiao
- SEAWAT: Misspelled DENSEREF in VDF dialog
- SEAWAT: Disable input option incorrectly
- SEAWAT: Parameters named incorrectly in VSC package dialog.
- Errors when import MODPATH solution. - User: Toru Itazu
- fnMapFunctionToGridXSections
- flow vector displayed in cell with all flow leaving through well - User: Richard Peralta
- Crash in MT3D areal source/sinks dialog
- make a test for mtiCheckRt3dChemica
- Double Precision MODFLOW/MT3D Bug - Christian Winde Pedersen
- crash after trying to open locked project
- GMS 8.2 beta hangs when mapping SFR2 coverage to MODFLOW - User: Bruce Campbell
- Cells look spiky in LAK package tutorial
- Vertical conductivity/vertical anisotropy units not updating - Shuangshuang Xie
- Import from web issues
- SEAWAT: Diable input trigger incorrectly in VDF dialog
- deleting TIN node doesn't delete value from dataset
- SEAWAT: Default coefficient values in VSC Package
- Zonebudget will not run

Intermediate Release GMS 8.1.7 - Feb 17, 2012

- Scatter points colored by data set value have limited range of -5 to 5 - User: Phillip C. de Blanc
- MNW with MT3D
- Source/Sink Limit not updating in MT3D - User: Eric Lappala
- Crash when opening project file - User: William Werner
- Stochastic Inverse Model doesn't run with more than one material set combination.
- GMS crashes when converting 3d mesh to 3d scatter
- Crash when deleting time step number
- Crash when exporting to levee analyst DB
- GMS 8.1 - MT3D / GCG solver input labels - User: Toru Itazu
- Converting Mesh to Scatter
- Line properties help
- "Tutfile" Tutorial Text Bug

- TIN contour labels do not show up until you create a new coverage and feature object - User:Calvin Erdman
- Unique values contour option crashes GMS - User: Thomas Mühlbauer
- crashes at end of MODFLOW Translator run - User: David Li
- crash when selecting multiple 3d mesh nodes - User: Lance Jones
- Problem opening “*.bas” file in GMS 8.1.6 32-Bit which opens in 8.1.6 64-bit - User: Mary Halstead
- select cross section and then delete from project explorer
- Starting head values aren't saved after running the model.

Intermediate Release GMS 8.1.6 - Dec 13, 2011

- MODFLOW translator problem
- Crash when setting up time series plot in plot wizard - User: Chaoying Jiao
- "Cannot open model output file..." error when running PEST with tutorial file
- grid to CAD data error
- Crash when changing the page orientation to landscape with PDFCreator set as default printer - User: Michael Popp
- Cannot disable arcobjects - User: Eric Lappala
- Problem with multiplier/dataset in Modflow parameters - User: Tim Cowdery
- Parameter estimation box in Modflow | Parameters does not remain checked after saving GMS
- time steps grayed out/unavailable in the 3D Data Set -> Grid window - User: Sean Czarniecki
- MODPATH polygon disappears
- crash when opening .gpr file - User: William Werner
- MODFLOW/MT3DMS do not run with 64-bit and parallel options selected - User:Eric Lappala

Intermediate Release GMS 8.1.5 - Nov 17, 2011

- Contours on iso-surfaces do not animate correctly
- Display of inactive cells
- block fill
- Intermediate image tests bug
- terraserver not working
- Kriging interpolation in tutorial not working properly
- Mixing up of polygon properties - User: Thomas Mühlbauer
- Improper display of refined points - User: Walid Sabbah
- modflow files do not read into GMS 8.1 - User: Jeff Davis
- Vector legend text should be white with black background
- When redrawing display, z magnification goes to 1
- Error in Generating data from solids tutorial
- Unexpected stranded cell warning
- Project file will not save with SEEP2D
- Scalar values in display options does not work
- GMS hangs when trying to load file - User: Chris Burke
- Inactive Cells not Displayed
- Switching to ortho mode changes layer
- After running MODFLOW flow translator, error reading in files, crash
- water capacity curve button does not work in FEMWATER materials window - User: Julian Weir

Intermediate Release GMS 8.1.4 – Sept 28, 2011

- Cell faces option doesn't work properly with inactive cells
- Display issues when the size of symbol attributes.
- GMS crashes when loading the starting file for SEAWAT conceptual approach tutorial
- 3D Grid data to Grid doesn't work
- GMS crashes when assigning upper value for iso-surface. - Geostatistics 3D tutorial.
- Line does not show up with single arrowhead
- FEMWATER saves mass related cards when it's a flow only simulation
- Progress bar when converting point shapes to scatter points
- Exporting shape from 2D scatter
- Linear contours do not disappear when turned off
- 64 bit: Enable ArcObjects delete all existing shape file data
- MODFLOW parallel 64-bit does not run with SAMG solver - User: Thomas Mühlbauer
- Select With Polyline command not enabled correctly
- GMS doesn't load the correct contour option display - related to bug 1252
- Community edition doesn't work until restart
- Times not saved in ADH Time and Output Control
- GMS 2D mode: Attribute table is not visible.
- GMS crashes when testing the save and reload of a shared test project.
- MODFLOW layers display doesn't get updated.
- SEEP2D Tutorial Issue
- SEEP2D Tutorial Issue
- Auto-Fill Blank Cross Section doesn't work correctly.
- 3D grid contour legend diplayed when 3D grid turned off.
- GlobalMapperInterface warning window pop up when importing shapefile/ArcObject data.
- GMS crashes when opening file
- GMS crashes when displaying both iso-surfaces and contours
- Error message when opening the project with existing MODPATH solution
- Cross Section Options window does not appear
- Vector color legend issue
- Text not visible when contours are turned on
- 2D grid cell numbers do not line up with the grid
- Display issues with block fill and color fill
- Crash when enabling specified data set contours
- Grid node number colors issue
- IJK triad text color issue
- Block fill contours not reflecting contour options
- GMS doesn't show inactive cells for 2D grid data - Image test
- GMS 8.1 doesn't automatically frame image after translation - User: Giep du Toit
- GMS 8.1 doesn't automatically frame image after translation - User: René J. Suárez-Soto
- Texture mapping weird banding
- Inactive cells do not display correctly
- GMS 8.1 - SEAM3D tutorial Crash
- Solids faces are fuzzy

Intermediate Release GMS 8.0.11 - Nov 9, 2011

- Vector legend text should be white with black background
- When redrawing display, z magnification goes to 1
- water capacity curve button does not work in FEMWATER materials window - User: Julian Weir

Intermediate Release GMS 8.0.10 - Sept 22, 2011

- Unselected polygons shown in polygon attributes window with show selected option active - User: Giep du Toit
- 3D Grid cell edges not displayed - User: Mary Southerland
- Warning pops up after computing particles with Boundary option turned on - User: Julian Weir
- GMS crashes after adding one more layer and saving the project file - User: Richard F. Carbonaro
- Z magnification doesn't work with IJK triad on
- MODFLOW simulation from GMS 6.0 does not load into GMS 8.0 - User: Diesner, Kai
- When editing 3D mesh nodes, a phantom node appears
- Capture Zone Polygon Disappears
- Fill should not be behind text
- Scatter set only partially visible
- When turning the CAD data off, the CAD layers remain checked
- Too many error messages when interpolating
- Enabling IJK triad changes z magnification
- Named layer ranges do not influence T-PROGS output - User: Pasqualino Littera
- Cannot create MODFLOW simulation from bounding 3D grid - User: Kuhn, Thomas
- Warning message when add image with arcobjects enabled
- Cannot select inactive cells in front or side ortho mode
- Max # sources and sinks isn't set correctly in MT3DMS model.
- Map to MODFLOW freezes at 60% for specific model - User: Clarkson, Maria
- Errors when trying to display element faces - User: Julian Weir
- GMS outputs files to MODFLOW that aren't specified in the project - User: Tim Cowdery
- Contour doesn't work correctly with starting heads.
- Change vertical anisotropy ratio input from Kh/Kv to Kv/Kh - User: Julian Weir
- MODFLOW runs in GMS but not in USGS MODFLOW - User: Xiulan He
- Freeze when loading GMS project file
- Data offset by 3 cells after running CCF -> Data Sets
- Inconsistent display of pathlines - User: Thomas Mühlbauer
- Polygon displayed when deleted
- Linear contour legend display issue
- Crash when changing the solution time step
- Crash when generate unsat curve in femwater
- Extra TIN data sets created when reprojecting
- Crash when scrolling through time steps
- FEMWATER manual incorrect on CB2, NB2 cards
- Crash when going to Ortho Mode
- GMS hangs after running SEEP2D.

Intermediate Release GMS 8.0.9 - Aug 9, 2011

- Crash when loading *.gpr file - User: Emanual Amen Storey
- Generating MODPATH pathlines takes up more than 2-3 GB of RAM - Jack Dahl
- Interpolate To | MODFLOW Layers command does not run successfully - User: Jack Dahl
- PEST does not run with MODFLOW 2005 - User: Julian Wier
- Problem with Network Hardware Lock with single license in GMS 8.0.7 - Julian Weir
- NEVTOP equals 3 is not available in the “MODFLOW EVT Package” window - User: Kirstin Neff
- Dates will not change in time series windows - User: Medhat El bihery
- “Export” item in MT3D menu should not be there - User: Rene J. Suarez-Soto
- Default observation target scale is 0.0 - User: Todd Wood
- Native MF2K files exported from GMS do not load back into GMS - User: Xiulan He
- Reopening project with HUF materials sets loses material IDs - User: Xiulan He
- 3D grid layer boundary display issue
- Breakpoint triggered - Risk Analysis Wizard Debug assertion failed
- Color is not updated in the Materials window and Layer Ranges Legend
- Cells edges are displayed wrong
- Crash when changing HGU in Borehole editor with attached cross section
- Color fill bug - default data set range - only happens to the computer with low-end NVIDIA graphic cards
- Color fill bug - transparency doesn't work - only happens to the computer with low-end NVIDIA graphic cards
- FEMWATER polygon properties mass flux units
- FEMWATER right-click menu properties command disabled
- FEMWATER spec. mass flux getting written as variable conc.
- FEMWATER symbols not dissappearing
- Frame Happens on Hitting Transform's Cancel Button
- Grid not appearing when project opens
- Hangs on Converting Map to 2D Mesh
- Import from web doesn't work on the advanced case study 1
- Indices for inactive cells are displayed
- Many error messages when animating
- Map to MT3D Recharge Crashes When No MF Recharge Package
- Model runs in 7.1 but not 8.0
- Named Layer Ranges inconsistencies
- Nothing happens when you select transparency
- Project does not close when you click the projects 'x' button in the upper right corner
- SEAWAT import error
- Screen Capture reverse the drawing order in GMS
- The whites of draped image are transparent

Intermediate Release GMS 8.0.8 - Jun 20, 2011

- Crash when opening GMS project file created in GMS 6.5.6 - User: Anne Waitz Vistrup
- Error when attempting to convert solids to modflow - User: Bruce Campbell
- Crash when enabling arcobjects - User: Chad Ballard
- Background color changes to black after importing shapefile through GIS module - User: Chaoying Jiao
- Crash when converting layer contours to arcs - User: Christoph Möbus
- LINK-AMG (LMG) Package control parameters incorrect - User: Christoph Möbus
- GMS 8.0 fails to run PEST with the parallel pest option turned on - User: Julian Weir
- Borehole data set
- CMainGraphicsWindow constructed but not deleted.
- Crash reading large modflow simulation
- Crash when editing borehole contact elevation
- Reproject scatter points doesn't work
- GMS doesn't automatically set the projection for the project based on the import data
- GMS needs to automatically reproject the importing data to the current projection.
- Image coordinate stays the same when you reproject a project
- Image import has no watch cursor at first
- Memory leak at exit in EnableDockingEx(CBRS_ALIGN_ANY);
- MODFLOW Display Options Check None doesn't change Wells (MNW1) and Drains (DRT1)
- Select all doesn't work with point
- Screen capture with obs targets
- Time value isn't listed properly for plotting

Intermediate Release GMS 8.0.7 – May 24, 2011

- TIN vertices “jump” down when you start dragging them in oblique view - User: Stefanie Lamb
- Contours incorrectly displayed when toggling on specify a range - User: Jeffrey Davis
- Problem with frame macro in GMS 8.0.6 - User: Julian Weir
- Grid merge and display Problems - User: Matt Gamache
- “Long Shadows” being displayed when displaying cell faces and inactive cells - User: Christoph Möbus
- Inactive cells not displayed correctly - User: Christoph Möbus
- *.sen file not saved correctly with DRT parameter while running PEST - User: Xiulan He
- KMZ animation with dates/times uses wrong times - User: Suarez-Soto, Rene J. (ATSDR/DHAC/EISAB)
- Animation with 3D Grid Vectors
- Crash when reproject global projection
- Contour Options dlg - legend toggle
- Resize Modflow Observation dialog
- Computed flow with CHOB arc groups
- Remove the following from shared flineutl.cpp since it uses definitions in GMS
- NUZTOP Default Value Incorrect in UZF Package
- Annotations: Anchoring not restored on save/reload
- Annotations: Image Properties dialog, color button dimming
- Annotations: Imported annotation using % isn't sized or positioned right
- Annotations: Properties dialog titles are inconsistent, as are tab names
- Annotations: Scale bar fills behind with the wrong background color
- Annotations: Anchor tab undoes Revert To Original Aspect Ratio
- Annotations: No way to have an image be its native size

- Improper Value in IUZFBND When Mapping to MODFLOW
- SFR package - DLEAK
- GMS kmz animation always saved in default directory, and avi is always saved
- Crash converting layer contours to arcs
- GMS Crashes on Selecting Material Zone Without Grid
- Crash On Material Edit After Deleting Cross-Section
- Export native mf2k text - Array Multiplier
- Flow vs Time plot
- Window named MODFLOW-2000 when running PESTusing MF2K5
- MODFLOW 2K5 doesn't run when using advanced parameter option
- Can't change text format for factor of safety in UTEXAS
- Can't change color for flow line in SEEPD2D
- files not opening correctly in GMS 8.0
- Legend in animation with 3D Grid Vectors
- Transient RCH data

Intermediate Release GMS 7.1.10 - March 2, 2011

- GMS 7.1 displaying slower then 6.5 - Andrzej Rodzoch
- False positive in MODFLOW model checker
- Mf stress periods dialog
- Importing mf simulation from state of UT
- Blanks Error message
- Problems running Pest - Bianca Pedersen
- modflow.bf file
- Rotate Tool not being unselected when ortho mode is turn on
- Transform Not working
- Not saving Multiplier for longitudinal Dispersivity - Frank Tsai
- Changing the Ibound in a polygon does not change make the cells inactive - Toni Smith
- Crash when Assigning ELEVUP
- computed flows not displaying - Julian Weir
- Create animation crashes GMS
- Insert 3D grid layer crash
- Gaussian Fields bug

Intermediate Release GMS 7.1.9 - January 12, 2011

- 32 bit crash when Auto-Assigning Horizons - Johan Haan
- Edit Projection Crash - Xiaoyong Zhan
- Stream flows
- Layer Contours -> Arcs cannot find existing linear contours
- Transform command on Cross Sections
- Map to WASH123D missing from project explorer right click menu - Jerry Lin
- Problems reading in the ETS Package - Mary Halstead
- Problems with the Contours legend - John Czarnecki
- Crash running simulation check - Rose McAndrew
- Mapping to MODFLOW overwrites data not in specified layer range - Ken Heim
- Nothing happens when right cliking on a coverage and selecting reproject

- Crash when mapping to MODFLOW - Julian Weir
- MODFLOW coverage areal properties layer range attribute
- Oblique view while in ortho mode
- Mesh is patching and now paving - Sittinan Benjasupattananan
- Problems with recharge being deleted - Mary Halstead
- Cannot hide the active coverage when hiding inactive coverages.
- Feature arc Z field functionality
- Mass vs. Time plot crashes when a polygon is selected.
- Problems running MT3D - Julian Weir
- Export arcs into shape file doesn't work.
- MODFLOW files not importing.
- Crash when opening the starting file for the lake package tutorial.

Intermediate Release GMS 7.1.8 - November 18, 2010

- MT3DMS computed mass flux is not reported.
- Fieldgen input was being written to the wrong location.
- Grid would shift when selecting a 2D Scatter Data set.
- Pasting into times dialog for "Print at specified times" would hang GMS.
- GIS layer files now use a relative path.
- Grid's display didn't get updated in normal mode.
- Couldn't merge 2D grid cells.
- Clock display in film loop didn't work.
- GMS didn't convert scatter data imported from web to the correct projection.

Intermediate Release GMS 7.1.7 - October 18, 2010

- Sometimes GMS would crash with an error message of "rows, columns, or layers do not match the current grid"
- The cross section display occasionally would not get updated when material color or pattern was changed.
- 2D Mesh was being paved instead of created as a patch.
- Clock animation wasn't displaying correctly. - John Czarneki
- GMS would crash after incorrectly parsing a project. - Bianca Pedersen
- Layer contours could show crazy values.
- Primary and secondary storage coefficients would not load when opening NAM file in GMS. - Jobst Massmann
- MODFLOW translator wouldn't ensure file names were unique. - Mary Halstead
- Reprojection would not always correctly apply to the vertical datum. - Michael Boiardi
- Porocity wouldn't update. - Philippe Perrier
- The Shift key when connection points didn't work correctly.
- GMS wouldn't report when there was a problem creating a 3D Mesh. - James Bellin
- GMS occasionally couldn't locate the MFR file.
- GMS would launch itself when encountering "Error Launching Model".
- Materials were wrong when importing from a GeoStudio file.
- Residuals in modflow output were calculated differently than the residuals in the attribute table. - Rene Suarez-Soto

Intermediate Release GMS 7.1.6 - August 19, 2010

- Contour Labels did not convert correctly. - John D. Gallinatti
- Z was displayed uner the wrong Project Explorer item.
- Cross Sections did not always match the Solids. - Matt O'Banion
- Google Earth Vector would include hidden feature points.
- Recharge wouldn't map to MT3D correctly. - Julian Weir
- Cross Section Editor would incorrectly display vertices.
- GMS would write the 2D Mesh file without E6T cards.
- GMS wouldn't recalculate the shading color start value.
- Texture mapped image wouldn't display correctly.
- GMS was not assigning materials when using Solids to MODFLOW Layers. - Bill Simpkins
- Printing would produce faint contours. - Jenny Malmquist
- GMS wouldn't save material properties correctly. - Julian Weir
- GMS would crash opening a file. - Daniel Meyer
- TIN contours were incorrect.
- Map -> MODFLOW with recharge would crash some 32-bit machines. - Julian Weir
- The legend wouldn't show the correct values with recharge. - Julian Weir
- The selected cell in a 3D Grid wouldn't display properly.
- Building a Horizon coverage could crash GMS
- Layered Mesh would display errors and not correct create a 3D Mesh. - James Bellin
- GMS 3.1 files wouldn't read into 7.1. - Reinhard Zapata
- Help button for SEEP2D Materials would display the wrong information.
- Fixed several SEEP2D New Simulation errors.

Intermediate Release GMS 7.1.5 - July 15, 2010

- Interpolating a scatter set to a layer would take a really long time. - Val Britton.
- Recharge would change the head more than expected. - Julian Weir
- Pasting information into the Stress Periods dialog would cause a crash.
- Fix problem with invalid NROW/NCOL in binary files.
- Modflow importer problem involved LMB solver data would cause GMS 7.1 to not read GMS 6.0 files.
- Refining 3D Mesh elements with a cross section would cause a crash.

Intermediate Release GMS 7.1.4 - June 28, 2010

- Memory issue when reading a particular project with last release. -Bruce Campbell
- Only able to display the computed flux for one FEMWATER node at a time. - Anders Soderstrom
- Coverages could not be rearranged or dropped into folders. - Kevin Winters
- New materials could be created with non-unique IDs when a materialwith ID 0 existed. - Michael Boiardi
- Increased speed on importing large MODFLOW models into GMS.
- Increased speed on reading and saving large MODFLOW models. Can increase speed by disabling option to compress MODFLOW h5 data in Edit | General Preferences. This causes more hard disk memory to be used.
- Was possible to cause GMS to crash by entering a small segment sizewhen redistributing arc vertices.
- Material patterns were not drawn correctly on the sides of a grid.

Intermediate Release GMS 7.1.3 - June 15, 2010

- Crash selecting confined radio button in HUF dialog.
- New coverages could appear as child objects of improper project explorer tree items. - Kevin Winters
- Seep2D simulation created even though cancel was selected.
- Creating a bounding 2D Grid could causes a crash.
- Turning on Draw tools and saving in a different location caused crash.
- HUF data sets outside HUF sim folder when reading 6.5 TPROGs HUF sample.
- Pasting larger amount of spreadsheet data into package dialog caused crash.
- Converting dataset to CAD with existing CAD data could cause crash.
- Graphics window does not update after hiding a 3D mesh element selected by clicking.
- Bad cross section extents caused "Convert to Coverage" to fail. - Ken Pattermann
- 2D Scatter -> 2D Mesh can result in 2 meshes.
- Inconsistent feature arcs to 3D mesh cross sections. - Barbara Donnell
- 2D mesh dataset duplication adds to dataset size when nodes have been deleted.
- Not able to move nodes in Cross Section Editor when node display off in Map Display Options.
- File Import Wizard does not ignore missing values in unmapped columns. - Barbara Donnell
- Merging cells with an MT3D simulation results in crash.
- MODFLOW translator fails when reading external binary array. - Shabbir Ahmed
- Unable to open MT3D GCG package dialog.
- MODAEM does not respond to changes in porosity. - Soheil Afshari
- The X, Y cursor location is not updated during process of creating a feature arc. - Kevin Winters
- When displaying vector arrows the view won't zoom, pan, or rotate. - Carl Davies
- The Unselect all tool doesn't update the display.
- Redistribute Vertices fills in bad default spacing and crashes with default. - Kevin Winters
- All row change shouldn't affect non-applicable cells in Attribute Table. - Kevin Winters
- Crash while reading gpr file with invalid map data.
- Plot axes in the graphics view don't pan or rotate.
- Mulitple MT3D plot issues. - Kevin Winters
- Drawing Tools don't display after drawing and not turned on when added to project explorer.
- Project not loaded after double clicking on it in Windows Explorer - David Blackmore
- Crash when reading project file with missing MODFLOW solution.
- Crash when reading project file with missing MT3D solution.
- Crash on importing species in MT3D BTN package dialog.
- Reading the modpath sample gives errors looking for files.
- Importing into an existing project could overwrite existing materials with imported materials without warning.
- Exporting 2D scatter points from tree item and module folder produces different results. - Jerry Lin
- Drag and drop the sample femwater fws and material project explorer item gets created in wrong place.
- Crash could happen when generating particles.
- Scatter Interpolation to Layer taking a really long time. - Val Britton
- Display final array values for pilot points not updating correctly.
- In some cases zone budget doesn't match USGS zone budget.
- Possibility of writing out of bounds when reading FEMWATER BC file.
- SEEP2D Attribute Table does not sort based on BC Type. - Kevin Winters
- Frame current cross section or all cross sections not working in Cross Section Editor.
- Reading in a UTEXAS solution crashes crash.
- Crash while loading project file.
- UTEXAS - When saving in GMS 2D mode it thinks an existing project should be saved as new project.

- UTEXAS - Factor of safety and solution doesn't display.
- UTEXAS - Fixed grid coordinate not saved.
- UTEXAS - Fixed grid display not correct.
- UTEXAS - Fixed grid option not exporting correctly.
- UTEXAS - Display options not saved and restored.
- UTEXAS - Distributed loads not displayed orthogonally.
- UTEXAS - Not exporting GRAPH line.
- UTEXAS - Distributed load arrows linger.
- UTEXAS - Only exporting one distributed load arc.

Intermediate Release GMS 7.1.2 - April 26, 2010

- The MODFLOW boundary conditions don't initially appear unless you uncheck and recheck MODFLOW. - Scott Morgan
- Material legend text randomly doesn't appear when copying to an image.
- Modflow translator crashing on import of external binary array. - Erick Powell
- Some wiki links to tutorials not working correctly.
- Converting shapes to feature objects not working. - Hisham Zarour
- WASH123D 1D mesh coordinate conversion not working. - Kevin Winters
- Crash during horizons to solids. - Eskelinen Anu
- MODFLOW Sources/Sinks shows incorrect K value.
- Use last somehow set for first stress period resulting in MODFLOW solutions not matching.
- A particular MODFLOW model crashes in 7.1 when running this model but in 6.5 works great. - Hal Davis
- Opening particular shape file results in incorrect background color and program hangs.
- Crash in 2D mode when creating points using attribute table.
- Modflow translator imports incorrect values for external binary array. - Erick Powell
- When reading 6.5 TPROG HUF sample data sets show up outside HUF sim folder.
- Error appears and pathlines not displayed upon generating particles. - Jeremy Robinson
- Print button not working in borehole cross section editor. - Grzegorz Jeleniewicz
- Sometimes hangs when generating contours.
- GMS hangs when loading RT3D files. - Ki Young Cha
- Grid dimensions are changed during transformation. - Julian Weir
- Zone budget IDs weren't mapping and saving correctly. - Julian Weir

Intermediate Release GMS 7.1.1 – March 15, 2010

- Unable to extract 2D water table from FEMWATER. - James Bellin
- Double-click on project (*.gpr) file doesn't load it in GMS.
- Unable to convert CCF file to data sets. - Griet Heuvelman
- Color fill won't display correctly for pore pressure in SEEP2D.
- Unable to import MODFLOW name file with spaces. - Donato Sollitto
- Error message appears when selecting different TIN after using Snap Vertices to TIN. - Jim Wieck
- Merging coverages does not carry the attributes.
- Crash on deleting selected nodes for mesh.
- Button in Shading Options Dialog doesn't handle window resize properly.
- Doesn't delete old SEEP2D simulation when creating a new one.
- No flow head of 999.99 changes to 999.98902 after MODFLOW import.
- MODFLOW translator not importing files. - Julian Weir

Intermediate Release GMS 7.0.4 - February 22, 2010

- Don't automatically run model checker.
- GMS crash when deleting MT3D or grid. - Bruce Siegmund
- Crash setting up new RT3D model with user defined reaction.

Intermediate Release GMS 7.0.3 - January 4, 2010

- Initial Concentrations Not mapping to MT3D.
- Max Rate of Substrate Utilization not Saved in Seam3d. - Mark Widdowson and Bruce Campbell
- Switching tools redraws the screen when it shouldn't.

Intermediate Release GMS 7.0.2 - October 29, 2009

- Convert to CAD Error. - Thomas Burke
- Project that opens in 6.5 but not 7.0. - John Czarnecki
- Crash when trying to interpolate 2D scatter to MODFLOW layers. - Nathan Haws
- GMS 7.0 Crash in MODFLOW Global Options with MT3D. - Nathan Dye
- FEMWATER | Geometry BUG. - Simon Woodward
- Some boreholes don't show on opening file.
- Van Genuchten parameters not saving.
- Duplicate vector data set.
- CCF time steps don't show times.
- Image doesn't clear switching from plan view to previous view. - Kevin Winters
- Blanks Warning when making a big 2D grid.
- Crash while create new species in MT3D.
- Open 2dg file crashes GMS 7.0.
- GMS Crash saving with empty RIV package. - Scott Morgan
- Display bugs in ortho mode.
- Bad interpolation results using interpolate scatter to mesh. - Kevin Winters
- Observed flow of arcs not displaying. - Val Britton
- Bad help button links. - Kevin Winters
- Borehole labels become frozen in place after screen capture. - Kevin Winters
- ADH Display Options tab title is superfluous. - Kevin Winters
- File Import Wizard crashes when no coverage data exists. - Kevin Winters
- Screen does not update after changing conceptual model types. - Kevin Winters
- Time step window shows incorrect dates for MODFLOW ET datasets. - Craig Altare
- GMS crash during 2D Data Set -> Layer. - Scott Morgan
- Unable to edit number of instances in Parameters dialog.
- Dry cells displayed with SFR2 sample.
- Bad use previous after map to MODFLOW for STR Package. - Craig Altare
- MT3D Select Data Set window is missing datasets and crashes. - Craig Altare
- MODFLOW ET Surface dataset values are displayed as zero. - Craig Altare
- ADH coverage properties not working properly. - Kevin Winters
- Failed to load particle tracking points from project file.
- Crash When turning on modflow contours. - Marielle Labadens
- Abort and Read solution doesn't work with MODFLOW.
- Texture mapping - image doesn't go away on new.
- File Import Wizard's Transient node data type is broken for WASH123. - Kevin Winters

- Incomplete MODFLOW STR Package file when exporting native MK2F files. - Craig Altare
- Model Checker Doesn't Display Selected Cells.
- No Sample files with Modpath tutorial. - Scott Morgan
- GMS not printing correctly. - Herman Wolfs
- Water Table Display not Working as it should. - Mark Greenwood
- Pasting point information into Feature Properties window is mishandled. - Kevin Winters
- GMS Save Error with FEMWATER. - Simon Woodward
- Flow budget difference between 6.5 and 7.0. - John Czarneki
- Delete Arcs crash. - Tim Cowdery
- Can't run MT3D even when enabled in certain cases. - Kim Gordon

Intermediate Release GMS 7.0.1 – September 9, 2009

- TSIM window title incorrect. - Kevin Winters
- Mapped materials changes from conceptual model not saved in project.
- Crash if using delete key to delete SEEP2D mesh.
- Crash when defining conceptual model species.
- CAD drawing issues when changing views.
- Setting manual contour options doesn't work.
- Lake flow budget not displayed.
- Pathlines sometimes not regenerated for MODAEM.
- Won't show selected cells in second layers if conceptual model displayed.
- Materials with patterns not displaying correctly in ortho mode.
- TIN edge color should be disabled when auto selected.
- Fixed error when using well screens and observations.
- Problems with mesh generation and display issues. - Kevin Winters
- Grid overlay with keq issue. - Sean Czarniecki
- Crash when importing conceptual model species. - Eric Lappala

Intermediate Release GMS 6.5.6 - May 29, 2009

- Map -> WASH123D incorrect flow rates. - Clarissa Hansen
- MODPATH not displaying arrows on path line. - Val Britton
- Extra rows in 2D/3D Grid Properties Dialogs.

Intermediate Release GMS 6.5.5 - April 22, 2009

- Gaussian MODFLOW simulations don't work.
- Bug when creating new cube and specifying center.
- MT3DMS recharge concentration change doesn't save.
- MT3D bug with calculate mass dialog.
- Update message for images covered by arc objects.
- Fixed help link to xmswiki for color ramp.

Intermediate Release GMS 6.5.4 - February 2, 2009

- Snap to Cross Section to Tin Crash. - Bill Simpkins
- Export Mf2k to native text. Error given "no text". - Bruce Campbell
- MT3D New Simulation command continuing after cancel.
- Map->MODFLOW with Transient Specified head not creating a step function. -Clarissa Hansen
- Dry wells symbol for MODAEM wells not working correctly

Intermediate Release GMS 6.5.3 - December 30, 2008

- Material edits not saved - David Zuccala
- Moving New Coverage
- Observation Points with 3D meshes
- GMS not saving borehole diameter
- 3D Grid properties incorrectly changing the Y and Z of the grid origin

Intermediate Release GMS 6.5.2 - October 17, 2008

- Map -> 2D Mesh doesn't map materials if "Meshing options" isn't on
- Material edits not saved - David Zuccala
- Minor issue with copying scatter sets - Clarissa Hansen
- GMS with ARCGIS Issues - Sean Czarniecki
- Image issues with Arc Objects enabled - Sean Czarniecki
- 3D Data Set to Grid does not always work properly - Christoph Möbus
- SEEP2D coverage with head option causes hang - Cary Talbot

Intermediate Release GMS 6.5.1 – September 9, 2008

- Import of Modflow 96 files - error reading the OC file
- Text import wizard crash when setting the text qualifier to 'none'
- Error saving out changes to recharge file
- Transient properties for lakes not mapping correctly
- RT3D preventing assignment of Mass loading bcs
- Iso-surface crash on 3D mesh with 5 node elements
- Crash when executing the Horizons->3D Mesh with a 2D mesh containing duplicate nodes
- UTEXAS tension crack info not written correctly
- Setting the numeric values on a color ramp based on the data from a 3D scatter set
- Converting shape file polylines to arcs where the polylines contain distint separate line segments
- Grid navigation arrowing displayed consistently
- Display of MT3D species concentration in the Point source/sink dialog
- Update to Gms2Mf2k.exe (Export native MF2K files) to handle filenames with spaces and not placing quotes around file names in the name file so that the USGS version of modflow is able to read the file

References

[1] http://www.aquaveo.com
[2] http://www.aquaveo.com/GMS-pricing
[3] http://www.aquaveo.com/downloads/gms

Dialog Help

This is a special page that relates GMS dialogs to wiki pages. GMS reads this page when a user hits the *Help* button in a GMS dialog, and opens the wiki at the page indicated below.

Wiki Page | Dialog Number | Dialog ID

1. Transform | 3544 | IDD_CHANGE_GEOMETRY
2. Projections#Reproject | 18297 | IDD_REPROJECTCURRENTDIALOG
3. Projections#Reproject | 18299 | IDD_REPROJECTSINGLEPOINTDIALOG
4. Projections | 18296 | IDD_CURRENTPROJECTIONDIALOG
5. DE4 Package | 22061 | IDD_MF_SOLVER_DE4
6. NWT Package | 22060 | IDD_MF_SOLVER_NWT
7. MODFLOW Translator | 22044 | IDD_MF_TRANSLATOR
8. Color Options | 6 | IDD_COLOROPTIONS
9. Point Display Attributes | 46 | IDD_POINT_DISPLAY_ATTS
10. New Palette | 80 | IDD_NEWPALETTE
11. PEST | 285 | IDD_MF_PARAMEST_PEST
12. MODFLOW Observations | 107 | IDD_MF_OBS_DLOG
13. HUF Package | 109 | IDD_MF_HUF
14. Borehole Editor | 129 | IDD_BH_EDITOR
15. Material Properties | 22040 | IDD_MATERIALs
16. Initialize MT3DMS Time Steps | 131 | IDD_MT_INIT_TIMESTEPS
17. GIS Tables | 137 | IDD_ARCGISTABLE
18. Layers | 138 | IDD_ARCGISLAYERS
19. GIS to Feature Objects | 140 | IDD_ARCGISMAPPING
20. MODFLOW Output Control | 143 | IDD_MF_OUTPUT
21. Select Data Set | 144 | IDD_SELECT_DATASET
22. Contour Label Options | 147 | IDD_CONTOUR_LABEL_OPTS
23. Observation Coverage | 148 | IDD_OBSERVE
24. Select by Location | 155 | IDD_ARCGISSELECTBYLOCATION
25. Selection Options | 157 | IDD_ARCGISSELECTIONOPTIONS
26. Global Options | 226 | IDD_MF_GLOBALPACKAGE
27. Stress Periods | 231 | IDD_INITIALIZE_TIMES
28. Stochastic Wrapper | 264 | IDD_STOWRAPPER
29. Dialog | 265 | IDD_PESTWRAPPER
30. Material ID Editor | 266 | IDD_MF_MATERIALSP
31. MODPATH Particle Sets | 268 | IDD_MP_PARTICLE_SETS
32. Threshold Analysis | 278 | IDD_WIZ_PPTHRESH2
33. Parameter Estimation Dialog | 285 | IDD_MF_PARAMEST_PEST
34. MODFLOW Source/Sink | 286 | IDD_MF_SOURCE_SINK
35. MODFLOW Sources/Sinks | 943 | IDD_MF_SS_TABLE
36. Borehole | 288 | IDD_DISPOPTS_BOREHOLE

37. Solid | 290 | IDD_DISPOPTS_SOLID
38. 2D Mesh | 292 | IDD_DISPOPTS_2DMESH
39. SEEP2D | 297 | IDD_DISPOPTS_2DMESH_SEEP2D
40. 2D Grid | 298 | IDD_DISPOPTS_2DGRID
41. Scatter Point Set | 300 | IDD_DISPOPTS_SCATTER
42. Viewing the Printed Output File | 301 | IDD_CHOOSETEXTEDITOR
43. 3D Mesh | 304 | IDD_DISPOPTS_3DMESH
44. Femwater | 305 | IDD_DISPOPTS_3DMESH_FEMWATER
45. ADH Display Options | 306 | IDD_DISPOPTS_3DMESH_ADH
46. Map | 308 | IDD_DISPOPTS_MAP
47. Save | 315 | IDD_SAVEDIALOG
48. Select From List | 316 | IDD_SELECT_FROM_LIST
49. TPROGS | 317 | IDD_TPROG_TSIM_IN
50. Cross Section | 320 | IDD_DISPOPTS_CROSS_SECTIONS
51. Get Online Password | 347 | IDD_ONLINEPASS
52. Enable GMS | 348 | IDD_ENABLE_BETA
53. Coordinate Transformations | 361 | IDD_COORD_CONVERT
54. User Defined Ellipsoid | 374 | IDD_COORD_ELLIPSE
55. Database Import Wizard | 382 | IDD_DATABASE_WIZ3
56. MODAEM | 388 | IDD_DISPOPTS_MODAEM
57. 3D Mesh Refinement Options | 391 | IDD_3DMESH_REFINEMENTOPTS
58. Stress Periods | 392 | IDD_MF_STRESSPER
59. Stress Periods | 393 | IDD_MF_STRESS_PERIODS_NEW
60. Conceptual Model Properties | 402 | IDD_CONMOD_PROP
61. Options | 404 | IDD_PLOT_OPTIONS
62. Coverage Setup | 405 | IDD_COV_SETUP
63. GMS:Feature Objects | 409 | IDD_FPROP
64. AMG1R5 Solver | 410 | IDD_MF_LMG_DISCLAIMER
65. WASH123D Global Options | 416 | IDD_WASH_GLOBAL_OPTS
66. WASH123D Chemicals | 417 | IDD_WASH_CHEMICALS
67. WASH123D 3D Flow | 419 | IDD_WASH_SOLVOPTS_3DFLOW
68. WASH123D 3D Transport | 421 | IDD_WASH_SOLVOPTS_3DTRANSPORT
69. WASH123D 2D Flow | 423 | IDD_WASH_SOLVOPTS_2DFLOW
70. WASH123D 3D Flow | 425 | IDD_WASH_ITER_PARAM_3DFLOW
71. WASH123D 3D Transport | 426 | IDD_WASH_ITER_PARAM_3DTRANSPORT
72. WASH123D 2D Flow | 427 | IDD_WASH_ITER_PARAM_2DFLOW
73. WASH123D 3D Flow | 428 | IDD_WASH_INIT_CONDS_3DFLOW
74. WASH123D 3D Transport | 429 | IDD_WASH_INIT_CONDS_3DTRANS
75. WASH123D 2D Flow | 430 | IDD_WASH_INIT_CONDS_2DFLOW
76. WASH123D Mass Tracking | 431 | IDD_WASH_MASS_TRCKING
77. WASH123D 3D Flow and Transport | 432 | IDD_WASH_OUTPUT_CONTROL_3D
78. WASH123D 3D Flow | 433 | IDD_WASH_OUTPUT_CONTROL_2DFLOW
79. WASH123D 3D flow | 434 | IDD_MATERIALS_WASH_3DFLOW
80. WASH123D 3D Transport | 435 | IDD_MATERIALS_WASH_3DTRANSPORT
81. WASH123D 2D Flow | 436 | IDD_MATERIALS_WASH_2DFLOW
82. WASH123D Parameter Iteration | 437 | IDD_WASH_ITER_PARAM_TAB
83. WASH123D Output Control | 438 | IDD_WASH_OUTPUT_CONTROL_TAB

84. WASH123D Solve Options | 439 | IDD_WASH_SOLVOPTS_TAB
85. WASH123D Initial Conditions | 440 | IDD_WASH_INIT_CONDS_TAB
86. WASH123D Time Control | 441 | IDD_WASH_TIME_CONTROL
87. Dialog | 442 | IDD_MF_SOURCE_SINK_TAB
88. 2D Mesh Polygon Properties | 454 | IDD_2DMESH_POLY_PROP
89. Stochastic Histogram | 469 | IDD_MF_STOHIST
90. Animation Wizard - Options | 470 | IDD_ANIM_WIZ_OPTS
91. Data Calculator | 471 | IDD_DATA_CALCULATOR
92. Basic Transport Package | 476 | IDD_MT_BASIC_PACKAGE
93. Coordinate Transformation Wizard - Step 1 | 480 | IDD_COORD_WIZ
94. Coordinate System | 482 | IDD_COORD_SYSTEM
95. WASH123D | 486 | IDD_DISPOPTS_3DMESH_WASH
96. Images | 487 | IDD_DISPOPTS_IMAGE
97. Fill Between TINs -> Solids | 488 | IDD_FILL_TINS2SOLIDS
98. Debug | 489 | IDD_DEBUG_NEW
99. Save Options | 490 | IDD_FILE_OPTIONS_IMG
100. Copy to Clipboard | 491 | IDD_COPY_SCALE
101. Create Sphere | 492 | IDD_CREATE_SPHERE
102. Create Cylinder | 493 | IDD_CREATE_CYLINDER
103. Create Cube | 494 | IDD_CREATE_CUBE
104. Create Prism | 495 | IDD_CREATE_PRISM
105. Register Image | 498 | IDD_REGIMAGE_GM2
106. XY Series Editor| 500 | IDD_XYSERIES
107. WASH123D 1D Flow | 503 | IDD_WASH_SOLVOPTS_1DFLOW
108. Fix Layer Errors | 504 | IDD_FIX_LAYERS
109. Layer Transparency | 506 | IDD_ARCGISTRANSPARENCY
110. Find Duplicate Nodes | 508 | IDD_DUP_OPTS
111. GMS | 510 | IDD_STANDARD_INPUT_LONG
112. GMS | 511 | IDD_STANDARD_INPUT
113. Extrude/Offset Tin -> Solid | 512 | IDD_TIN_EXTRUDE
114. Dialog | 513 | IDD_TIN_FILL
115. GMS:Calculate Mass | 514 | IDD_MT_CALCMASS
116. Gaussian Simulation Options | 515 | IDD_INTERP_GAUSS
117. FEMWATER Time Control | 517 | IDD_FW_TIMECONTROL
118. FEMWATER Run Options | 518 | IDD_FW_RUNOPTIONS
119. GMS:The Display Menu | 520 | IDD_EDIT_VIEW
120. Network Lock Settings | 522 | IDD_NETWORKLOCK
121. Welcome to GMS | 523 | IDD_ENABLE_OR_DEMO
122. Hardware Lock | 525 | IDD_HARDWARE_LOCK
123. Security String | 526 | IDD_SECURITY_STRING
124. Network Lock Settings | 527 | IDD_NETWORK_SETTINGS
125. Arc -> Cross Sections | 528 | IDD_POLYLINE_TO_CROSS_SECTION
126. Register | 529 | IDD_REGISTER
127. GMS:Converting 3D Grids to Other Data Types#MODFLOW Layers -> 2D Scatter Points | 530 | IDD_MF_LAYERS_TO_SCATTER_NEW
128. Variable BC Options | 531 | IDD_FW_VARBCOPTS
129. Initialize Time Steps | 532 | IDD_FW_INITTIMESTEP

130. GMS:SEAM3D | 533 | IDD_NAPLDISSPACKAGE
131. Edit Properties | 540 | IDD_GENERIC_PROPERTIES
132. GMS:SEAM3D | 542 | IDD_SEAM3D_COMETPAC
133. Build Horizon Coverages | 543 | IDD_BLD_HOR_COV1
134. MODFLOW Lake Package | 593 | IDD_MF_LAKEPACKAGE
135. Display Themes | 598 | IDD_DISP_TEMPLATES
136. Plot Axes Selected Axis Properties | 638 | IDD_AXES_OPTS
137. GMS | 643 | IDD_METADATA_FIELD
138. GMS | 656 | IDD_YESNOCANCEL
139. Offset Arc | 679 | IDD_ARCOFFSET
140. Stitch Meshes | 680 | IDD_STITCH_MESH
141. Auto-assign Horizons | 698 | IDD_AUTO_HOR
142. Auto Select | 734 | IDD_AUTOFILL
143. Sample Data -> Stratigraphy | 738 | IDD_BH_SAMPLE_DATA_TO_STRAT
144. Borehole Cross Section Editor | 740 | IDD_BHXSECT_SNAPTIN
145. Web Service Authentication | 749 | IDD_WEBSERVICE_GETTOKEN
146. Feature Objects | 750 | IDD_SELFEATUREOBJ
147. Create Velocity Data Set | 752 | IDD_CREATE_VELOCITY_DS
148. ADH Constituent Material Properties | 106 | IDD_ADH_CONSTITUENT_MATERIAL_PROPERTIES
149. ADH Constituents | 105 | IDD_ADH_CONSTITUENTS
150. ADH Output Control | 103 | IDD_ADH_OUTPUT_CONTROL
151. ADH Analysis Options | 763 | IDD_ADH_ANALYSIS_OPTS
152. ADH Analysis Options | 101 | IDD_ADH_ANALYSIS_OPTIONS
153. Create Node/Face/Edge Set | 764 | IDD_ADH_CREATE_SET
154. Face Sets | 766 | IDD_ADH_FACE_SETS
155. Boundary Conditions | 768 | IDD_ADH_BOUND_CONDS
156. Node Properties | 769 | IDD_ADH_NODE_PROPS
157. View Sets | 771 | IDD_ADH_VIEW_SETS
158. Map -> 2D Mesh | 905 | IDD_SNAPMAP2DMESH
159. TIN Settings | 1027 | IDD_VERTEX_OPTIONS
160. Auto Select | 1033 | IDD_BH_AUTO_SEL
161. Set Operations | 1041 | IDD_SET_OPERATIONS
162. MODFLOW Multi-Node Well Package | 1049 | IDD_MF_SS_MNW
163. MODFLOW Multi-Node Well (MNW2) Package | 22064 | IDD_MF_SS_MNW2
164. MNW2 Wells | 4498 | IDD_MF_MNW2_WELLS
165. FEMWATER Geometry and Model Title Cards | 1051 | IDD_FW_TITLES
166. Building Solids and 3D Meshes with TINs | 1052 | IDD_3D_MESH_CREATE
167. Create Finite Difference Grid | 1055 | IDD_CREATE_GRID
168. 2D Mesh Settings | 1061 | IDD_NODE_OPTS
169. FEMWATER Initial Conditions | 1063 | IDD_FW_INITIAL_CONDITIONS
170. FEMWATER Iteration Parameters | 1078 | IDD_FW_ITER
171. Triangulation | 1096 | IDD_TRI_SUBDIV
172. Properties | 1098 | IDD_ELEM_MAT
173. SIP Package | 1128 | IDD_MF_SIP_PACKAGE
174. Text Properties | 1132 | IDD_TEXT_ATTS
175. SSOR Package | 1139 | IDD_MF_SSOR_PACKAGE
176. PCG2 Package | 1140 | IDD_MF_PCG2_PACKAGE

177. 3D Data -> 2D Data | 1146 | IDD_3DDATASET_TO_2DDATASET
178. Converting Borehole Data To Other Data Type | 1156 | IDD_BH_SAMPLE_DATA_TO_SCATTER_PTS
179. Interpolate Dialog | 1158 | IDD_INTERP_TO_OBJ
180. Relax Element Options | 1160 | IDD_RELAX
181. Advection Package | 1172 | IDD_MT_ADVECTION
182. Advection Package | 1174 | IDD_MT_PARTS
183. Dispersion Package | 1175 | IDD_MT_DISPERSION_PACKAGE
184. Importing Borehole Data | 1186 | IDD_IMPORT_SAMPLE_DATA
185. IFACE | 1217 | IDD_MP_IFACE
186. MODPATH ITOP | 1219 | IDD_MP_ITOP
187. General Options | 1232 | IDD_MP_GENERAL_OPTIONS
188. Find Point | 1242 | IDD_SELECT_SCAT
189. Map -> 2D Mesh | 1243 | IDD_MAP_TO_2DMESH
190. Map -> Model | 1245 | IDD_MAP_TO_MODEL
191. Clean | 1253 | IDD_CLEAN_FEATURE_OBJECTS
192. Node BC | 1256 | IDD_SEEP2DBC
193. SEEP2D Analysis Options | 1257 | IDD_SEEP2D_ANALYSIS
194. Horizontal Flow Barriers | 1269 | IDD_TOGGLE_FLOW_BARRIER
195. Import USGS DEMs | 1271 | IDD_DEM_IMPORT
196. Interpolate to MODFLOW Layers | 1277 | IDD_MAP_DATA_NEW
197. Generate Pressure Head Initial Conditions | 1334 | IDD_FW_GENERATE_IC
198. GCG Solver | 1345 | IDD_MT_GCG_SOLVER
199. Breakline Options | 1350 | IDD_BREAKLINE_OPTIONS
200. Elec. Acc. | 1353 | IDD_SEAM3D_BIODEG_ELEC_ACC
201. Gen. Coeff. | 1354 | IDD_SEAM3D_BIODEG_GEN_COEFF
202. Min. Conc. | 1355 | IDD_SEAM3D_BIODEG_MIN_CONC
203. Rates | 1357 | IDD_SEAM3D_BIODEG_RATES
204. Sat. Conc. | 1358 | IDD_SEAM3D_BIODEG_SAT_CONST
205. Use Coeff. | 1359 | IDD_SEAM3D_BIODEG_USE_COEFF
206. Start Conc. | 1372 | IDD_SEAM3D_BIODEG_START_CONC
207. Geometry File | 1373 | IDD_FW_GEOM_FILE
208. Solids -> MODFLOW | 4704 | IDD_SOLIDTOMODLOW
209. Jacknifing Summary | 1415 | IDD_JACKKNIFE_SUMMARY
210. Concentrations | 1437 | IDD_SEAM3D_DECHLOR_CONC
211. Saturation Constants | 1438 | IDD_SEAM3D_DECHLOR_SAT_CONST
212. Rates | 1439 | IDD_SEAM3D_DECHLOR_RATES
213. Coefficients | 1440 | IDD_SEAM3D_DECHLOR_COEFFS
214. Reductive Dechlorination Factors | 1441 | IDD_SEAM3D_DECHLOR_RED_DEC_FACT
215. Model Wrapper | 1464 | IDD_MODELWRAPPEROPTS
216. Interpolation Options | 1467 | IDD_INTERP_OPTS
217. 2D IDW Interpolation Options | 1468 | IDD_IDW_OPTS
218. Subset Definition | 1472 | IDD_SUBSET_PROP
219. Natural Neighbor Options | 1473 | IDD_NATNEIGH_OPTS
220. Kriging Options | 1474 | IDD_KRIG_OPTS
221. Variogram Editor | 1475 | IDD_VARIOGRAM_EDITOR
222. Search Ellipsoid | 1476 | IDD_KRIG_SEARCH_ELLIPS
223. Experimental Variogram | 1477 | IDD_VARIOGRAM_OPTS

224. Drift Coefficients | 1478 | IDD_KRIG_DRIFT
225. Search Options | 1479 | IDD_KRIG_SEARCH_OPTS
226. Lighting Options | 1482 | IDD_DISPOPTS_LIGHTING_OPTIONS
227. MODFLOW Packages | 1483 | IDD_MF_PACKAGES
228. Materials Transport | 1484 | IDD_MATERIALS_FACT_TRANSPORT
229. FEMWATER Flow | 1485 | IDD_MATERIALS_FEMWATER_FLOW
230. FEMWATER Transport | 1486 | IDD_MATERIALS_FEMWATER_TRANSPORT
231. MODFLOW | 1487 | IDD_MATERIALS_MODFLOW
232. SEEP2D | 1488 | IDD_MATERIALS_SEEP2D
233. Redistribute Vertices | 1489 | IDD_REDISTVERT
234. van Genuchten Curve Generator | 1490 | IDD_CURVE_GENERATOR
235. Data -> DXF | 1544 | IDD_DATA_TO_DXF
236. Set Window Boundaries | 3340 | IDD_WINDOWBOUNDS
237. Lighting Options | 3343 | IDD_SHADINGOPTS
238. Pattern Properties | 3350 | IDD_PATTERN_DISPLAY_ATTS
239. Mesh -> 3D Tets | 3352 | IDD_MESH_3DTETS
240. Color Table | 3356 | IDD_PALETTE_TABLE
241. Plot Wizard Step 1 of 2 | 3357 | IDD_PLOTWIZ1
242. Plot Wizard Step 2 of 2 | 3358 | IDD_PLOTWIZ2
243. Reference Time | 3359 | IDD_REFTIME
244. General | 429253114 | XMS_CONTOURS_IDD_CONTOURS_SPECIFIC
245. Grid Layers | 3361 | IDD_CONTOURS_GRIDLAYER
246. Edit Data Set Values | 3362 | IDD_EDIT_DATASET
247. Active/Inactive Flags | 3363 | IDD_ACTIVE_INACTIVE_FLAGS
248. MODFLOW Evapotranspiration Package | 1429155828 | IDD_MF_AREAL_PACK
249. MODFLOW ETS Package | 1142990693 | IDD_MF_ETPACKAGE
250. Areal Source/Sink | 3365 | IDD_MT_AREALSS
251. Observed Flux Options | 3366 | IDD_FLUXPROPS
252. 3D Grid | 3367 | IDD_DISPOPTS_3DG_GRID
253. MODFLOW | 3368 | IDD_DISPOPTS_3DG_MODFLOW
254. MT3D | 3369 | IDD_DISPOPTS_3DG_MT3D
255. Particles | 3370 | IDD_DISPOPTS_3DG_PARTICLES
256. RT3D Chemical Reaction Package | 3371 | IDD_RT_CHEMICAL_REACTION
257. Define Species | 3372 | IDD_MT_DEFINE_SPECIES
258. TIN | 3373 | IDD_DISPOPTS_TIN
259. T-PROGS Options | 3374 | IDD_MARKOVOPTS
260. Scatter Data Color Options | 3375 | IDD_DATA_COLOR_OPTIONS
261. Observation Points -> Scatter Points | 3376 | IDD_OBS2SCATPTS
262. Generate Particles | 3377 | IDD_MP_GEN_PARTICLES
263. Tree | 3378 | IDD_TREE
264. Print Print Options | 3379 | IDD_PRINTSETUP2
265. Line Properties | 3390 | IDD_LINEARROW_ATTS
266. GMS | 3391 | IDD_RECTOVAL_ATTS
267. TPROGS | 3395 | IDD_TPROGS_MODEL
268. Variable Time Steps | 3397 | IDD_MT_VAR_TIMESTEPS
269. Scatter Point Set Options | 3399 | IDD_SCAT_DATASETOPTS
270. MT3D/RT3D Packages | 3400 | IDD_MT_PACKAGES

271. Define Species | 3401 | IDD_SEAM3D_DEFINE_SPECIES
272. GMS | 3402 | IDD_ABOUT
273. FEMWATER | 3404 | IDD_FW_MODEL
274. Redistribute Layers | 3408 | IDD_3DGRD_REDISTLAYS
275. Data Set Info | 3410 | IDD_DATASET_INFO1
276. GMS | 3411 | IDD_MATERIALS_VS2D
277. Stochastic Inverse Model | 3412 | IDD_MF_STOINVMODEL
278. View Values | 3417 | IDD_PLOT_VIEWVALUES
279. Iso-Surface Volumes | 3418 | IDD_ISOSURF_VOLUMES
280. Properties | 3424 | IDD_AEM_FPROP
281. GMS | 3531 | IDD_METADATA_MAIN
282. GMS | 3532 | IDD_METADATA_PROFILE
283. GMS | 3533 | IDD_METADATA_SOURCE
284. GMS | 3535 | IDD_METADATA_SPATIAL
285. Line Properties | 4045 | IDD_LINE_DISPLAY_ATTS
286. Vertical (Z) Markov Chain | 4101 | IDD_MARKOVCHAINS
287. LPF Package | 4102 | IDD_MF_LPF
288. GMG Solver | 4103 | IDD_MF_GMGSOLVER
289. MODFLOW Well Package | 1249798983 | IDD_MF_SS
290. MODFLOW Drain Package | 996857831 | IDD_MF_SS
291. MODFLOW Drain Return Package | 4133 | IDD_MF_SS
292. MODFLOW River Package | 10943839 | IDD_MF_SS
293. MODFLOW General Head Package | 1419895336 | IDD_MF_SS
294. MODFLOW Time Variant Specified Head Package | 1774843214 | IDD_MF_SS
295. Initialize Stress Period | 4137 | IDD_INIT_STRESSPERIOD
296. MODFLOW Stream Package | 4530 | IDD_MF_STREAMPACKAGE
297. MODFLOW Recharge Package | 435688408 | IDD_MF_AREAL_PACK
298. Source/Sink Mixing Package | 4146 | IDD_MT_MIXING_PACKAGE
299. Flux Options | 4150 | IDD_FLUXOPTS
300. Axis Titles | 4202 | IDD_AXISTITLES
301. Model Wrapper | 4203 | IDD_MODELWRAPPER
302. Stochastic Options | 4224 | IDD_MF_STOCHASTICOPTS
303. GMS | 4225 | IDD_MATERIALS_FACT_FLOW
304. Spreadsheet | 4227 | IDD_MF_SPREADSHEET
305. Units | 4228 | IDD_UNITS
306. Times | 4230 | IDD_TIMES
307. Solids -> HUF | 4253 | IDD_SOLIDTOHUF
308. GMS | 4254 | IDD_PROPERTIES
309. Generate Particles at Wells | 4269 | IDD_MP_GEN_PARTICLES_AT_WELLS
310. Cells | 4270 | IDD_FLOW_BUDGET_CELL
311. Block Centered FLow Package | 4271 | IDD_MF_BCF
312. Reading Stochastic Solutions | 4272 | IDD_STO_READSOL
313. Dialog | 4274 | IDD_STOMT3DWRAPPER
314. Run Options | 4275 | IDD_MT_RUN_OPTIONS
315. Risk Analysis Wizard | 4276 | IDD_WIZ_PP1
316. Capture Zone Analysis | 4277 | IDD_WIZ_PPCAP2
317. Scatter Data Color Options | 4281 | IDD_SCAT_COLOR_OPTS

318. Vector Options | 4282 | IDD_VECTOR_OPTIONS
319. Iso-surfaces Options | 4283 | IDD_ISO_SURFACES_OPTIONS
320. Cell Shading Options | 4284 | IDD_CELL_SHADING_OPTIONS
321. MODPATH Errors | 4302 | IDD_SHOW_TEXT
322. Flow Trace Options | 4322 | IDD_FLOW_TRACE_OPTS
323. Stochastic MODFLOW 2000 | 4328 | IDD_MF_STOMODEL
324. PEST | 4329 | IDD_MF_PESTMODEL
325. MODFLOW 2000 | 4343 | IDD_MF_MODEL
326. Select CAD Layers | 4344 | IDD_DXF_SELECT_LAYERS
327. Dialog | 4345 | IDD_MT_MODEL
328. Seep2d | 4346 | IDD_SP_MODEL
329. Material Set Info | 4350 | IDD_MATSET_INFO
330. MT3DMS | 4353 | IDD_MT_SELECTSOL
331. X-Axis Precision | 4354 | IDD_HISTOGRAM_PREC
332. File Import Wizard | 4359 | IDD_TXTWIZ1
333. File Import Wizard | 4360 | IDD_TXTWIZ2
334. Convergence Options | 4363 | IDD_MF_CONVERGENCE
335. Coordinate Systems | 4373 | IDD_COORDS
336. Coordinate Conversions | 4375 | IDD_COORD_TRANSFORMATION
337. MODFLOW - Horizontal Flow Barrier Package | 4376 | IDD_HFB_PACKAGE
338. Export Shapefile | 4377 | IDD_EXPORT_SHAPEFILE
339. GMS | 4379 | IDD_FONT_FLOAT
340. Database Import Wizard | 4380 | IDD_DATBASE_WIZ1
341. Database Import Wizard | 4381 | IDD_DATABASE_WIZ2
342. GMS | 4386 | IDD_QUERYFILE_OPTS
343. MODAEM Global Options | 4387 | IDD_AEM_GLOBAL
344. GMS | 4472 | IDD_COMBOBOX
345. Coordinate Conversion | 4473 | IDD_COORD_SINGLEPT
346. Plot Wizard | 4474 | IDD_PLOTWIZ3
347. Nonlinear | 4475 | IDD_WASH_SOLVOPTS_NONLINEAR
348. Wash123D | 4476 | IDD_WS_MODEL
349. Join Table | 4477 | IDD_GISGENERICJOIN
350. Animation Wizard - Cross Sections / Iso-Surfaces | 4478 | IDD_ANIM_WIZ_XSECTS
351. Animation Wizard - Data Sets | 4479 | IDD_ANIM_WIZ_DATA_SETS
352. Animation Wizard - Flow Trace | 4480 | IDD_ANIM_WIZ_FLOW_TRACE
353. Animation Wizard - Views | 4481 | IDD_ANIM_WIZ_VIEWS
354. Animation Wizard - Particle Tracking | 4482 | IDD_ANIM_WIZ_PARTICLES
355. Output Control | 4483 | IDD_MT_OUTPUT_CONTROL
356. Symbol Attributes | 4484 | IDD_SYMBOL_DISPLAY_ATTS
357. Reports | 4486 | IDD_REPORTS
358. Grid Frame | 4488 | IDD_GRIDFRAME
359. Modaem | 4489 | IDD_AEM_MODEL
360. GMS | 4490 | IDD_SELECT_TREE
361. Shape Files| 4491 | IDD_DISPOPTS_GIS
362. Plot Wizard - Step 2 of 2| 4492 | IDD_PLOT_FLOWBUDGET
363. Tree| 4493 | IDD_TREE1
364. DEM Smoothing Options | 4494 | IDD_DEM_SMOOTHING_OPTS

365. FEMWATER Output Control | 4499 | IDD_FW_OUTPUTCONTROL
366. FEMWATER Particle Tracking | 4501 | IDD_FW_PARTICLETRACKING
367. Transport Observation Package | 4502 | IDD_MT_TOBS
368. MT3DMS Chemical Reaction Packages | 4503 | IDD_MT_CHEMICAL_REACTION_PKG
369. Borehole Segment Properties | 4504 | IDD_BH_SOIL
370. Properties | 4505 | IDD_MT_AREAL_SOURCE_SINK
371. Parameters | 4507 | IDD_MF_PARAMS
372. Build Horizon Coverages | 4509 | IDD_BLD_HOR_COV3
373. Build Horizon Coverages | 4510 | IDD_BLD_HOR_COV2
374. Point Source/Sink BC | 4511 | IDD_MT_POINT_SOURCE_SINK
375. Build Solids | 4514 | IDD_HOR_INTERP_OPTS
376. Display Options | 4516 | IDD_DISPLAY_OPTIONS
377. Images | 4517 | IDD_DISPOPTS_IMAGE1
378. GMS | 4519 | IDD_MESSAGE_BOX_ASK_AGAIN
379. Drawing Grid | 4520 | IDD_DISPOPTS_DRAWING_GRID
380. Cross Section Editor | 4521 | IDD_BH_XSECT
381. Horizons Editor | 4522 | IDD_BH_HORIZONS
382. Print Setup | 4523 | IDD_PRINTSETUP3
383. Print Setup | 4524 | IDD_PRINTSETUP
384. Axes Display | 4525 | IDD_DISPOPTS_AXIS
385. XYZF Edit Bar | 4526 | IDD_EDIT_XYZF_BAR
386. MODFLOW Areal Package | 4527 | IDD_MF_AREAL_PACK
387. Open | 4528 | IDD_OPENDIALOG
388. Model Checker | 4529 | IDD_MODEL_CHECKER
389. Horizon Optimizer | 4532 | IDD_HOR_MODEL
390. Timestep | 4533 | IDD_TIMESTEP
391. Flow Budget | 4534 | IDD_FLOW_BUDGET
392. Zones | 4535 | IDD_FLOW_BUDGET_ZONE
393. File Import Filter Options | 4536 | IDD_FILE_IMPORT_FILTER_OPTS
394. Find Feature Object | 4537 | IDD_FIND_FEATURE_OBJECT
395. Preferences | 4539 | IDD_PREFS
396. General Preferences | 4540 | IDD_PREFS_GENERAL
397. Model Preferences | 4541 | IDD_PREFS_MODELS
398. #Images_.2F_CAD/CAD Preferences | 4543 | IDD_PREFS_IMAGES_CAD
399. Horizons to Solids - Top and Bottom Elevation | 4544 | IDD_HOR_TOPBOT
400. Horizon Elevation | 4546 | IDD_HOR_INPUTS
401. Build Solids | 4547 | IDD_HOR_BUILDSOLID
402. Build Mesh | 4548 | IDD_HOR_BUILDMESH
403. TIN Preferences | 4549 | IDD_PREFS_TINS
404. 1D Mesh | 4550 | IDD_DISPOPTS_1DMESH
405. Borehole Preferences | 4551 | IDD_PREFS_BHS
406. Scatter Set Preferences | 4552 | IDD_PREFS_SS
407. Plot Wizard - Step 1 of 2 | 4553 | IDD_PLOT_FLOW
408. WASH123D | 4554 | IDD_DISPOPTS_2DMESH_WASH
409. Horizons to Solids - Top and Bottom Elevations | 4555 | IDD_HOR_TOPBOT_MESH
410. 2D Mesh Preferences | 4556 | IDD_PREFS_2DMESH
411. WASH123D | 4557 | IDD_DISPOPTS_1DMESH_WASH

412. WASH123D 1D Flow | 4560 | IDD_WASH_OUTPUT_CONTROL_1DFLOW
413. Conveyance Pipe Linkages | 4561 | IDD_WASH_MAGIC_PIPE
414. Create Contaminant Flux Data Set | 4566 | IDD_CREATE_CONTAMFLUX_DS
415. Create a Vector Data Set | 4567 | IDD_SCALAR_FUNCS_TO_VEC_FUNC
416. LINK-AMG (LMG) Package | 4685 | IDD_MF_LMGSOLVER
417. Save Options | 4686 | IDD_FILE_OPTIONS_XMDF
418. Model Checker | 4687 | IDD_MODEL_CHECKER1
419. Node Sets | 4693 | IDD_ADH_NODE_SETS
420. Edge Sets | 4694 | IDD_ADH_EDGE_SETS
421. X-Axis | 4695 | IDD_PLOT_OPTIONS_XAXIS
422. Y-Axis | 4696 | IDD_PLOT_OPTIONS_YAXIS
423. Data Service Options | 4703 | IDD_DATASERVICE_OPTIONS
424. Solids -> ADH | 1378 | IDD_SOLIDTOADH
425. UTEXAS Materials | 4712 | IDD_MATERIALS_UTEXAS
426. UTEXAS | 1612878051 | IDD_UTX_OPTIONS
427. UTEXAS Display Options | 4714 | IDD_DISPOPTS_UTEXAS
428. UTEXAS | 4715 | IDD_UTX_MODEL
429. GeoStudio Import | 1260444303 | IDD_SELECT_TREE
430. Classify Material Zones | 4564 | IDD_CLASSIFY_MATERIAL_ZONES
431. GMS:Parameter Estimation Dialog#SVD (Singular Value Decomposition) | 522625753 | IDD_PROPERTIES
432. GMS:Parameter Estimation Dialog#SVD-Assist | 7003907 | IDD_PROPERTIES
433. UZF Package | 963 | IDD_MF_UZF_PACKAGE
434. SFR Package | 22038 | IDD_MF_SS_SFR2
435. SEAWAT Global Options | 962| IDD_SEAWAT_GLOBAL_OPTIONS
436. VDF Package | 22050 | IDD_SEAWAT_VDF_PACK
437. VSC Package | 22051 | IDD_SEAWAT_VSC_PACK
438. GMS:Cell Properties | 472 | IDD_3DGRID_CELL_PROPERTIES
439. FEMWATER Fluid Properties | 22049 | IDD_PROPERTIES_FW_FLUID
440. GMS:Creating_a_TIN | 1365642895 | IDD_PROPERTIES
441. GMS:T-PROGS | 22053 | IDD_TPROGS_OPTIONS_BOREHOLES
442. GMS:Global_Options/Basic_Package#Confining Beds | 935 | IDD_MF_LAYCBD
443. GMS:Find_Grid_Cell | 1195 | IDD_FIND_NODE

Article Sources and Contributors

What is GMS? *Source*: http://www.xmswiki.com/xms/index.php?oldid=41425 *Contributors*: Tdye, Unjedai

The GMS Screen *Source*: http://www.xmswiki.com/xms/index.php?oldid=46327 *Contributors*: Jcreer, Ksmith

Tool Palettes *Source*: http://www.xmswiki.com/xms/index.php?oldid=41258 *Contributors*: Ksmith, Unjedai

Project Explorer *Source*: http://www.xmswiki.com/xms/index.php?oldid=34917 *Contributors*: Ksmith

Tutorials *Source*: http://www.xmswiki.com/xms/index.php?oldid=47824 *Contributors*: Alanlemon, Ksmith, Unjedai

64 bit *Source*: http://www.xmswiki.com/xms/index.php?oldid=33881 *Contributors*: Unjedai

License Agreement *Source*: http://www.xmswiki.com/xms/index.php?oldid=47410 *Contributors*: Jcreer, Ksmith

Registering GMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=37301 *Contributors*: Ksmith, Unjedai

Community Edition *Source*: http://www.xmswiki.com/xms/index.php?oldid=36781 *Contributors*: Ksmith

Graphics Card Troubleshooting *Source*: http://www.xmswiki.com/xms/index.php?oldid=47401 *Contributors*: Jcreer, Ksmith

Reporting Bugs *Source*: http://www.xmswiki.com/xms/index.php?oldid=36306 *Contributors*: Unjedai

The File Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=49394 *Contributors*: Jcreer, Ksmith, Unjedai

The Edit Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=36811 *Contributors*: Jcreer, Ksmith, Unjedai

Units *Source*: http://www.xmswiki.com/xms/index.php?oldid=36775 *Contributors*: Ksmith, Unjedai

Preferences *Source*: http://www.xmswiki.com/xms/index.php?oldid=38611 *Contributors*: Bdolinar, Ksmith

Materials *Source*: http://www.xmswiki.com/xms/index.php?oldid=45300 *Contributors*: Alanlemon, Ksmith

Material Set *Source*: http://www.xmswiki.com/xms/index.php?oldid=34927 *Contributors*: Ksmith

Coordinate Systems *Source*: http://www.xmswiki.com/xms/index.php?oldid=41675 *Contributors*: Jcreer, Ksmith

Coordinate Conversions *Source*: http://www.xmswiki.com/xms/index.php?oldid=37513 *Contributors*: Ksmith

Projections *Source*: http://www.xmswiki.com/xms/index.php?oldid=49174 *Contributors*: Jcreer, Ksmith, Unjedai

CPP Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=37717 *Contributors*: Ksmith

Geographic Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=36074 *Contributors*: Ksmith, Unjedai

Local Coordinate System *Source*: http://www.xmswiki.com/xms/index.php?oldid=38290 *Contributors*: Jcreer, Ksmith

Coordinate Transformation *Source*: http://www.xmswiki.com/xms/index.php?oldid=37910 *Contributors*: Ksmith, Wood

Transform *Source*: http://www.xmswiki.com/xms/index.php?oldid=36585 *Contributors*: Ksmith, Unjedai

The Display Menu *Source*: http://www.xmswiki.com/xms/index.php?oldid=36815 *Contributors*: Jcreer, Ksmith, Unjedai

Contour Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39633 *Contributors*: Jcreer, Ksmith, Unjedai

Animations *Source*: http://www.xmswiki.com/xms/index.php?oldid=45126 *Contributors*: Jcreer, Ksmith, Unjedai

Color Ramp *Source*: http://www.xmswiki.com/xms/index.php?oldid=34676 *Contributors*: Ksmith

Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=35628 *Contributors*: Ksmith, Unjedai

Drawing Grid Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=34680 *Contributors*: Ksmith

Vectors *Source*: http://www.xmswiki.com/xms/index.php?oldid=39648 *Contributors*: Alanlemon, Jcreer, Ksmith

Lighting Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=46709 *Contributors*: Cdickson, Jcreer, Ksmith

Plot Axes *Source*: http://www.xmswiki.com/xms/index.php?oldid=34682 *Contributors*: Ksmith

Annotations *Source*: http://www.xmswiki.com/xms/index.php?oldid=41245 *Contributors*: Jcreer, Ksmith

CAD Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39649 *Contributors*: Jcreer, Ksmith

Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=39650 *Contributors*: Jcreer, Ksmith

Data Sets *Source*: http://www.xmswiki.com/xms/index.php?oldid=39640 *Contributors*: Jcreer, Ksmith

Data Calculator *Source*: http://www.xmswiki.com/xms/index.php?oldid=46733 *Contributors*: Jcreer, Jdhowlett, Ksmith

Display Theme *Source*: http://www.xmswiki.com/xms/index.php?oldid=34651 *Contributors*: Ksmith

XY Series Editor *Source*: http://www.xmswiki.com/xms/index.php?oldid=30324 *Contributors*: Jdhowlett

Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=37729 *Contributors*: Ksmith, Tdye

Interpolation Commands *Source*: http://www.xmswiki.com/xms/index.php?oldid=39652 *Contributors*: Jcreer, Ksmith, Unjedai

3D Interpolation Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39653 *Contributors*: Jcreer, Ksmith, Unjedai

Steady State vs. Transient Interpolation *Source*: http://www.xmswiki.com/xms/index.php?oldid=39654 *Contributors*: Jcreer, Tdye

Linear *Source*: http://www.xmswiki.com/xms/index.php?oldid=39655 *Contributors*: Jcreer, Ksmith, Unjedai

Inverse Distance Weighted *Source*: http://www.xmswiki.com/xms/index.php?oldid=32569 *Contributors*: Htran

Shepards Method *Source*: http://www.xmswiki.com/xms/index.php?oldid=37691 *Contributors*: Ksmith

Gradient Plane Nodal Functions *Source*: http://www.xmswiki.com/xms/index.php?oldid=37731 *Contributors*: Ksmith, Unjedai

Quadratic Nodal Functions *Source*: http://www.xmswiki.com/xms/index.php?oldid=39656 *Contributors*: Jcreer, Ksmith

Subset Definition *Source*: http://www.xmswiki.com/xms/index.php?oldid=34939 *Contributors*: Ksmith

Computation of Interpolation Weights *Source*: http://www.xmswiki.com/xms/index.php?oldid=37736 *Contributors*: Ksmith

Clough-Tocher *Source*: http://www.xmswiki.com/xms/index.php?oldid=37747 *Contributors*: Ksmith

Natural Neighbor *Source*: http://www.xmswiki.com/xms/index.php?oldid=37750 *Contributors*: Ksmith, Unjedai

Kriging *Source*: http://www.xmswiki.com/xms/index.php?oldid=37763 *Contributors*: Ksmith

Kriging Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=34688 *Contributors*: Ksmith

3D Kriging *Source*: http://www.xmswiki.com/xms/index.php?oldid=34690 *Contributors*: Ksmith

Variogram Editor *Source*: http://www.xmswiki.com/xms/index.php?oldid=39658 *Contributors*: Jcreer, Ksmith

Jackknifing *Source*: http://www.xmswiki.com/xms/index.php?oldid=39659 *Contributors*: Jcreer, Ksmith, Unjedai

TIN Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=34951 *Contributors*: Ksmith

Creating a TIN *Source*: http://www.xmswiki.com/xms/index.php?oldid=39661 *Contributors*: Jcreer, Jdye, Ksmith

Editing a TIN *Source*: http://www.xmswiki.com/xms/index.php?oldid=39662 *Contributors*: Jcreer, Ksmith, Unjedai

TIN Settings *Source*: http://www.xmswiki.com/xms/index.php?oldid=39664 *Contributors*: Jcreer, Tdye

TIN Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39665 *Contributors*: Jcreer, Ksmith, Unjedai

TIN Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=30784 *Contributors*: Tdye

Converting TINS to Other Data Types *Source*: http://www.xmswiki.com/xms/index.php?oldid=39666 *Contributors*: Bdolinar, Jcreer, Ksmith, Unjedai

Building Solids and 3D Meshes with TINs *Source*: http://www.xmswiki.com/xms/index.php?oldid=39667 *Contributors*: Jcreer, Ksmith, Unjedai

Triangulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=34959 *Contributors*: Ksmith

TIN Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=47669 *Contributors*: Jcreer, Tdye

Boreholes Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=39700 *Contributors*: Htran, Jcreer, Unjedai

Creating and Editing Boreholes *Source*: http://www.xmswiki.com/xms/index.php?oldid=38382 *Contributors*: Ksmith, Tdye, Unjedai

Borehole Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39702 *Contributors*: Jcreer, Ksmith, Unjedai

Borehole Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=38598 *Contributors*: Ksmith, Tdye

Borehole Hydrogeologic Units *Source*: http://www.xmswiki.com/xms/index.php?oldid=39703 *Contributors*: Jcreer, Jdye, Unjedai

Converting Borehole Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=39704 *Contributors*: Jcreer, Ksmith, Tdye, Unjedai

Borehole Cross Sections *Source*: http://www.xmswiki.com/xms/index.php?oldid=37319 *Contributors*: Bdolinar, Ksmith, Unjedai

Horizons *Source*: http://www.xmswiki.com/xms/index.php?oldid=38634 *Contributors*: Alanlemon, Ksmith, Unjedai

Horizon Conceptual Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=39705 *Contributors*: Jcreer, Ksmith, Unjedai

Horizons Applications *Source*: http://www.xmswiki.com/xms/index.php?oldid=36737 *Contributors*: Ksmith, Unjedai

Horizons to HUF *Source*: http://www.xmswiki.com/xms/index.php?oldid=39707 *Contributors*: Alanlemon, Jcreer, Ksmith, Unjedai

Horizons Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=41263 *Contributors*: Alanlemon, Jcreer, Ksmith, Unjedai

Horizons to Solids *Source*: http://www.xmswiki.com/xms/index.php?oldid=39710 *Contributors*: Alanlemon, Jcreer, Ksmith, Unjedai

Horizons to 3D Mesh *Source*: http://www.xmswiki.com/xms/index.php?oldid=39711 *Contributors*: Alanlemon, Jcreer, Ksmith, Tdye, Unjedai

Solid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=39712 *Contributors*: Jcreer, Ksmith

Solid Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=39715 *Contributors*: Jcreer, Jdye

Solid Primitives *Source*: http://www.xmswiki.com/xms/index.php?oldid=35964 *Contributors*: Ksmith, Wood

Solid Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39716 *Contributors*: Jcreer, Ksmith, Wood

Solid Module Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=29402 *Contributors*: Unjedai

Solids to Layered Mesh *Source*: http://www.xmswiki.com/xms/index.php?oldid=39717 *Contributors*: Jcreer, Ksmith

Solids to HUF *Source*: http://www.xmswiki.com/xms/index.php?oldid=39718 *Contributors*: Jcreer, Ksmith

Solids to MODFLOW Command *Source*: http://www.xmswiki.com/xms/index.php?oldid=39721 *Contributors*: Jcreer, Ksmith

2D Mesh Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=34992 *Contributors*: Ksmith

Creating a 2D Mesh *Source*: http://www.xmswiki.com/xms/index.php?oldid=39725 *Contributors*: Bdolinar, Jcreer, Ksmith

Editing 2D Meshes *Source*: http://www.xmswiki.com/xms/index.php?oldid=39726 *Contributors*: Jcreer, Ksmith, Unjedai

2D Mesh Settings *Source*: http://www.xmswiki.com/xms/index.php?oldid=36006 *Contributors*: Ksmith, Tdye

2D Mesh Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39727 *Contributors*: Jcreer, Jdye, Ksmith

2D Mesh Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=36008 *Contributors*: Ksmith, Unjedai

Converting a 2D Mesh to other types of Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=39728 *Contributors*: Jcreer, Ksmith, Tdye, Unjedai

Element types *Source*: http://www.xmswiki.com/xms/index.php?oldid=39813 *Contributors*: Jcreer, Unjedai

2D Mesh Polygon Attributes *Source*: http://www.xmswiki.com/xms/index.php?oldid=39729 *Contributors*: Jcreer, Ksmith

2D Grid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=37591 *Contributors*: Ksmith, Unjedai

2D Grid Types *Source*: http://www.xmswiki.com/xms/index.php?oldid=30581 *Contributors*: Tdye

Creating and Editing 2D Grids *Source*: http://www.xmswiki.com/xms/index.php?oldid=38647 *Contributors*: Bdolinar, Jcreer, Ksmith, Unjedai

2D Grid Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39732 *Contributors*: Jcreer, Ksmith, Tdye, Unjedai

2D Grid Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=37593 *Contributors*: Tdye, Unjedai

Converting 2D Grids *Source*: http://www.xmswiki.com/xms/index.php?oldid=39733 *Contributors*: Jcreer, Ksmith, Tdye, Unjedai

2D Scatter Point Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=41804 *Contributors*: Bdolinar, Jcreer, Ksmith, Unjedai

Creating and Editing 2D Scatter Point Sets *Source*: http://www.xmswiki.com/xms/index.php?oldid=38644 *Contributors*: Bdolinar, Jcreer, Ksmith, Unjedai

2D Scatter Point Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39739 *Contributors*: Jcreer, Jdye, Ksmith, Unjedai

2D Scatter Point Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=30778 *Contributors*: Jdye

Interpolating with 2D Scatter Points *Source*: http://www.xmswiki.com/xms/index.php?oldid=38366 *Contributors*: Jdye, Ksmith, Unjedai

Converting 2D Scatter Points to Other Types of Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=39740 *Contributors*: Alanlemon, Jcreer, Tdye, Unjedai

Gaussian Field Generator *Source*: http://www.xmswiki.com/xms/index.php?oldid=39741 *Contributors*: Jcreer, Ksmith

Active/Inactive Points *Source*: http://www.xmswiki.com/xms/index.php?oldid=39743 *Contributors*: Jcreer, Tdye

2D Interpolation Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39744 *Contributors*: Jcreer, Ksmith

3D Mesh Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=34999 *Contributors*: Ksmith

3D Element Types *Source*: http://www.xmswiki.com/xms/index.php?oldid=35000 *Contributors*: Ksmith

Creating a 3D Mesh *Source*: http://www.xmswiki.com/xms/index.php?oldid=39746 *Contributors*: Jcreer, Ksmith, Unjedai

Editing a 3D Mesh *Source*: http://www.xmswiki.com/xms/index.php?oldid=39747 *Contributors*: Jcreer, Ksmith, Unjedai

3D Mesh Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39748 *Contributors*: Jcreer, Ksmith, Tdye

3D Mesh Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=35418 *Contributors*: Ksmith

Classify Material Zones *Source*: http://www.xmswiki.com/xms/index.php?oldid=38563 *Contributors*: Alanlemon, Unjedai

Iso Surfaces *Source*: http://www.xmswiki.com/xms/index.php?oldid=45398 *Contributors*: Alanlemon, Jcreer, Ksmith

Converting 3D Meshes to Other Data Types *Source*: http://www.xmswiki.com/xms/index.php?oldid=39751 *Contributors*: Jcreer, Ksmith, Tdye

Building the 3D Mesh from the FEMWATER Conceptual Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=39752 *Contributors*: Jcreer, Ksmith

3D Grid Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=35967 *Contributors*: Ksmith, Unjedai

3D Grid Types *Source*: http://www.xmswiki.com/xms/index.php?oldid=39754 *Contributors*: Jcreer, Ksmith

Creating and Editing 3D Grids *Source*: http://www.xmswiki.com/xms/index.php?oldid=48658 *Contributors*: Bdolinar, Jcreer, Ksmith, Morelandtom, Unjedai

3D Grid Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39756 *Contributors*: Jcreer, Ksmith

3D Grid Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=39757 *Contributors*: Jcreer, Ksmith, Wood

Classify Material Zones *Source*: http://www.xmswiki.com/xms/index.php?oldid=38563 *Contributors*: Alanlemon, Unjedai

3D Grid Viewing Modes *Source*: http://www.xmswiki.com/xms/index.php?oldid=36087 *Contributors*: Ksmith, Tdye

Converting 3D Grids to Other Data Types *Source*: http://www.xmswiki.com/xms/index.php?oldid=39758 *Contributors*: Jcreer, Ksmith

Exporting Grids *Source*: http://www.xmswiki.com/xms/index.php?oldid=39759 *Contributors*: Jcreer, Unjedai

Cell Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=44522 *Contributors*: Alanlemon, Ksmith

Active/Inactive Cells *Source*: http://www.xmswiki.com/xms/index.php?oldid=39760 *Contributors*: Jcreer, Ksmith

Named Layer Ranges *Source*: http://www.xmswiki.com/xms/index.php?oldid=36094 *Contributors*: Ksmith, Unjedai

Redistribute Grid Cells *Source*: http://www.xmswiki.com/xms/index.php?oldid=36095 *Contributors*: Ksmith, Unjedai

Redistribute Layers *Source*: http://www.xmswiki.com/xms/index.php?oldid=36096 *Contributors*: Ksmith, Unjedai

3D Scatter Point Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=41805 *Contributors*: Bdolinar, Ksmith

3D Scatter Point Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=41816 *Contributors*: Bdolinar, Jcreer, Ksmith, Tdye

3D Scatter Point Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=36099 *Contributors*: Ksmith, Tdye

Interpolating with 3D Scatter Points *Source*: http://www.xmswiki.com/xms/index.php?oldid=41830 *Contributors*: Bdolinar, Ksmith, Tdye

Converting 3D Scatter Points to Other Data Types *Source*: http://www.xmswiki.com/xms/index.php?oldid=41829 *Contributors*: Bdolinar, Jcreer, Jdye, Ksmith, Unjedai

Bounding Grid *Source*: http://www.xmswiki.com/xms/index.php?oldid=41827 *Contributors*: Bdolinar, Ksmith, Tdye

Map Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=35048 *Contributors*: Ksmith

Feature Objects *Source*: http://www.xmswiki.com/xms/index.php?oldid=44547 *Contributors*: Alanlemon, Jcreer, Ksmith

Conceptual Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=39766 *Contributors*: Jcreer, Unjedai

Feature Object Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=39767 *Contributors*: Bdolinar, Jcreer, Ksmith

Feature Object Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=39768 *Contributors*: Jcreer, Ksmith, Tdye

Coverages *Source*: http://www.xmswiki.com/xms/index.php?oldid=39769 *Contributors*: Jcreer, Ksmith, Unjedai, Wood

Grid Frame *Source*: http://www.xmswiki.com/xms/index.php?oldid=39770 *Contributors*: Jcreer, Ksmith, Tdye

Clean Command *Source*: http://www.xmswiki.com/xms/index.php?oldid=39771 *Contributors*: Jcreer, Tdye

Temporal Discretization *Source*: http://www.xmswiki.com/xms/index.php?oldid=41010 *Contributors*: Jcreer, Ksmith, Unjedai

Map to Models *Source*: http://www.xmswiki.com/xms/index.php?oldid=39773 *Contributors*: Djordan, Jcreer, Ksmith

Map to Modules *Source*: http://www.xmswiki.com/xms/index.php?oldid=39775 *Contributors*: Alanlemon, Jcreer, Ksmith

GIS Module *Source*: http://www.xmswiki.com/xms/index.php?oldid=28640 *Contributors*: Htran

GIS Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=30773 *Contributors*: Tdye

GIS Tool Palette *Source*: http://www.xmswiki.com/xms/index.php?oldid=36200 *Contributors*: Ksmith, Tdye

Enabling ArcObjects *Source*: http://www.xmswiki.com/xms/index.php?oldid=36201 *Contributors*: Ksmith

GIS to Feature Objects *Source*: http://www.xmswiki.com/xms/index.php?oldid=41441 *Contributors*: Alanlemon, Jcreer, Ksmith, Unjedai

Add Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=30770 *Contributors*: Tdye

Arc Hydro Groundwater *Source*: http://www.xmswiki.com/xms/index.php?oldid=36203 *Contributors*: Alanlemon, Ksmith

FEMWATER *Source*: http://www.xmswiki.com/xms/index.php?oldid=30977 *Contributors*: Unjedai

Building a FEMWATER Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=38600 *Contributors*: Ksmith

FEMWATER Conceptual Model Approach *Source*: http://www.xmswiki.com/xms/index.php?oldid=38666 *Contributors*: Bdolinar, Ksmith

FEMWATER Flows *Source*: http://www.xmswiki.com/xms/index.php?oldid=37090 *Contributors*: Ksmith, Unjedai

Point Source/Sink BC *Source*: http://www.xmswiki.com/xms/index.php?oldid=37089 *Contributors*: Ksmith, Tdye

Assign Node/Face BC *Source*: http://www.xmswiki.com/xms/index.php?oldid=37091 *Contributors*: Jdye, Ksmith

FEMWATER Point / Arc Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=38736 *Contributors*: Ksmith

FEMWATER Model Input *Source*: http://www.xmswiki.com/xms/index.php?oldid=37095 *Contributors*: Ksmith

FEMWATER Initial Conditions *Source*: http://www.xmswiki.com/xms/index.php?oldid=37096 *Contributors*: Ksmith, Wood

Saving a FEMWATER Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=37295 *Contributors*: Jschultze, Ksmith

FEMWATER Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37098 *Contributors*: Ksmith, Tdye

FEMWATER Post Processing Viewing Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37099 *Contributors*: Ksmith, Unjedai

FEMWATER Particle Sets *Source*: http://www.xmswiki.com/xms/index.php?oldid=33351 *Contributors*: Unjedai

MODAEM *Source*: http://www.xmswiki.com/xms/index.php?oldid=38347 *Contributors*: Djordan, Ksmith, Unjedai

MODAEM License Agreement *Source*: http://www.xmswiki.com/xms/index.php?oldid=30829 *Contributors*: Jdye

MODAEM Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37101 *Contributors*: Ksmith, Wood

MODFLOW *Source*: http://www.xmswiki.com/xms/index.php?oldid=48940 *Contributors*: Alanlemon, Bdolinar, Htran, Ksmith, Unjedai

MODFLOW 2005 *Source*: http://www.xmswiki.com/xms/index.php?oldid=35931 *Contributors*: Bdolinar, Ksmith

MODFLOW Packages Supported in GMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=38708 *Contributors*: Alanlemon, Bdolinar, Ksmith, Unjedai

BCF Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=43966 *Contributors*: Ksmith, Unjedai

CHD Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=35065 *Contributors*: Ksmith

DE4 Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=35692 *Contributors*: Alanlemon, Unjedai

DRN Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=34256 *Contributors*: Ksmith

DRT Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37105 *Contributors*: Ksmith

ETS Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37106 *Contributors*: Ksmith

EVT Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37109 *Contributors*: Ksmith

GHB Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37110 *Contributors*: Ksmith, Unjedai

Global Options/Basic Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37117 *Contributors*: Ksmith

GMG Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=36567 *Contributors*: Ksmith, Tdye

HFB Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37120 *Contributors*: Ksmith

HUF Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37121 *Contributors*: Ksmith

LAK Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=34268 *Contributors*: Ksmith

LMG Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37927 *Contributors*: Htran, Ksmith

LMG1 Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=34272 *Contributors*: Ksmith

LPF Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=43967 *Contributors*: Ksmith, Unjedai

MNW1 Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37125 *Contributors*: Bdolinar, Ksmith

MNW2 Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37126 *Contributors*: Bdolinar, Ksmith

NWT Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=35929 *Contributors*: Alanlemon, Bdolinar

PCG2 Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=34264 *Contributors*: Ksmith

RCH Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37127 *Contributors*: Ksmith

PEST ASP Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37472 *Contributors*: Ksmith, Unjedai

RIV Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37128 *Contributors*: Ksmith

SIP Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=34262 *Contributors*: Ksmith

SSOR Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=34266 *Contributors*: Ksmith

STR/SFR Packages *Source*: http://www.xmswiki.com/xms/index.php?oldid=41260 *Contributors*: Ksmith, Unjedai

UZF Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37134 *Contributors*: Htran, Ksmith

WEL Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=37135 *Contributors*: Bdolinar, Ksmith

Building a MODFLOW Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=31689 *Contributors*: Bdolinar

MODFLOW Array Editor *Source*: http://www.xmswiki.com/xms/index.php?oldid=37136 *Contributors*: Ksmith, Wood

MODFLOW Grid Approach *Source*: http://www.xmswiki.com/xms/index.php?oldid=44549 *Contributors*: Alanlemon, Bdolinar, Ksmith

MODFLOW Conceptual Model Approach *Source*: http://www.xmswiki.com/xms/index.php?oldid=44551 *Contributors*: Alanlemon, Ksmith

Activate Cells in Coverage *Source*: http://www.xmswiki.com/xms/index.php?oldid=37142 *Contributors*: Ksmith

Stress Periods *Source*: http://www.xmswiki.com/xms/index.php?oldid=37143 *Contributors*: Ksmith

Map to MODFLOW *Source*: http://www.xmswiki.com/xms/index.php?oldid=48663 *Contributors*: Alanlemon, Ksmith, Unjedai, Wood

Defining the Layer Elevations *Source*: http://www.xmswiki.com/xms/index.php?oldid=37792 *Contributors*: Ksmith

Standard MODFLOW Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=39690 *Contributors*: Jcreer, Ksmith

MODFLOW Parameters - Two Methods *Source*: http://www.xmswiki.com/xms/index.php?oldid=31263 *Contributors*: Unjedai

Conductance *Source*: http://www.xmswiki.com/xms/index.php?oldid=48739 *Contributors*: Ksmith, Unjedai

MODFLOW Parameters Disclaimer *Source*: http://www.xmswiki.com/xms/index.php?oldid=31262 *Contributors*: Unjedai

Saving and Reading a MODFLOW Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=37171 *Contributors*: Ksmith, Tdye

Output Control *Source*: http://www.xmswiki.com/xms/index.php?oldid=37151 *Contributors*: Ksmith

Importing MODFLOW Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=42193 *Contributors*: Alanlemon, Ksmith, Twood, Wood

MODFLOW with HDF5 *Source*: http://www.xmswiki.com/xms/index.php?oldid=37159 *Contributors*: Bdolinar, Ksmith

Unsupported MODFLOW Features *Source*: http://www.xmswiki.com/xms/index.php?oldid=37162 *Contributors*: Alanlemon, Ksmith

Interpolate to MODFLOW Layers *Source*: http://www.xmswiki.com/xms/index.php?oldid=41264 *Contributors*: Alanlemon, Ksmith

MODFLOW Layers to 2D scatter points *Source*: http://www.xmswiki.com/xms/index.php?oldid=37167 *Contributors*: Ksmith, Unjedai

MODFLOW Source/Sink Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=34701 *Contributors*: Ksmith

MODFLOW Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=34342 *Contributors*: Ksmith

MODFLOW Post Processing Viewing Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37800 *Contributors*: Alanlemon, Ksmith, Unjedai

MODFLOW Solution Properties Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=35093 *Contributors*: Ksmith

Saving and Reading a MODFLOW Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=37171 *Contributors*: Ksmith, Tdye

Cell Properties *Source*: http://www.xmswiki.com/xms/index.php?oldid=44522 *Contributors*: Alanlemon, Ksmith

MODFLOW World File *Source*: http://www.xmswiki.com/xms/index.php?oldid=37173 *Contributors*: Bdolinar, Ksmith, Unjedai

Error Summary Plot *Source*: http://www.xmswiki.com/xms/index.php?oldid=37174 *Contributors*: Ksmith

Zone Budget *Source*: http://www.xmswiki.com/xms/index.php?oldid=36888 *Contributors*: Ksmith, Unjedai, Wood

ZONEBUDGET *Source*: http://www.xmswiki.com/xms/index.php?oldid=36598 *Contributors*: Alanlemon, Ksmith

Viewing Computed Fluxes *Source*: http://www.xmswiki.com/xms/index.php?oldid=37176 *Contributors*: Ksmith, Wood

MODPATH *Source*: http://www.xmswiki.com/xms/index.php?oldid=38331 *Contributors*: Jdye, Morelandtom, Unjedai

MODPATH Particle Tracking *Source*: http://www.xmswiki.com/xms/index.php?oldid=37177 *Contributors*: Ksmith, Tdye, Unjedai

MODPATH Particle Sets *Source*: http://www.xmswiki.com/xms/index.php?oldid=37178 *Contributors*: Ksmith, Unjedai

MODPATH Zone Codes *Source*: http://www.xmswiki.com/xms/index.php?oldid=37179 *Contributors*: Ksmith, Tdye

MODPATH Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37184 *Contributors*: Ksmith, Unjedai, Wood

Exporting Pathlines *Source*: http://www.xmswiki.com/xms/index.php?oldid=43396 *Contributors*: Unjedai

MT3DMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=44540 *Contributors*: Alanlemon, Ksmith, Unjedai

MT3D Packages *Source*: http://www.xmswiki.com/xms/index.php?oldid=44530 *Contributors*: Alanlemon, Ksmith

MT3D Source/Sink Mixing Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=44539 *Contributors*: Alanlemon, Ksmith

Basic Transport Package Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=44524 *Contributors*: Alanlemon, Ksmith, Tdye

ICBUND Array *Source*: http://www.xmswiki.com/xms/index.php?oldid=44535 *Contributors*: Alanlemon, Bdolinar, Ksmith

MT3D Source/Sink Mixing Package *Source*: http://www.xmswiki.com/xms/index.php?oldid=44539 *Contributors*: Alanlemon, Ksmith

MT3D point Sources/Sinks BC *Source*: http://www.xmswiki.com/xms/index.php?oldid=44536 *Contributors*: Alanlemon, Ksmith

Building an MT3DMS Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44533 *Contributors*: Alanlemon, Ksmith

Saving an MT3DMS Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=44543 *Contributors*: Alanlemon, Tdye

Importing an MT3D Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=37201 *Contributors*: Alanlemon, Ksmith, Unjedai

MT3DMS Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=44541 *Contributors*: Alanlemon, Ksmith, Tdye

Calculate Mass *Source*: http://www.xmswiki.com/xms/index.php?oldid=44525 *Contributors*: Alanlemon, Ksmith

PEST *Source*: http://www.xmswiki.com/xms/index.php?oldid=46429 *Contributors*: Alanlemon, Ksmith

PHT3D *Source*: http://www.xmswiki.com/xms/index.php?oldid=47826 *Contributors*: Alanlemon, Jcreer

RT3D *Source*: http://www.xmswiki.com/xms/index.php?oldid=38340 *Contributors*: Jdye, Ksmith, Unjedai

Basic Transport Package Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=44524 *Contributors*: Alanlemon, Ksmith, Tdye

SEAM3D *Source*: http://www.xmswiki.com/xms/index.php?oldid=38346 *Contributors*: Jdye, Ksmith, Unjedai

SEAWAT *Source*: http://www.xmswiki.com/xms/index.php?oldid=38695 *Contributors*: Bdolinar, Ksmith

Building a SEAWAT Model *Source*: http://www.xmswiki.com/xms/index.php?oldid=38689 *Contributors*: Bdolinar, Ksmith

SEAWAT Global Options Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=38691 *Contributors*: Bdolinar, Ksmith

SEAWAT MODFLOW AUX Variables *Source*: http://www.xmswiki.com/xms/index.php?oldid=38688 *Contributors*: Bdolinar, Ksmith

SEAWAT Packages *Source*: http://www.xmswiki.com/xms/index.php?oldid=38692 *Contributors*: Bdolinar, Ksmith

SEAWAT Importing a SEAWAT Simulation *Source*: http://www.xmswiki.com/xms/index.php?oldid=38696 *Contributors*: Bdolinar, Ksmith

SEEP2D *Source*: http://www.xmswiki.com/xms/index.php?oldid=37213 *Contributors*: Ksmith

SEEP2D Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37803 *Contributors*: Ksmith, Tdye

SEEP2D Solution *Source*: http://www.xmswiki.com/xms/index.php?oldid=30366 *Contributors*: Unjedai

T-PROGS *Source*: http://www.xmswiki.com/xms/index.php?oldid=39781 *Contributors*: Jcreer, Ksmith, Unjedai, Wood

Vertical Markov Chain *Source*: http://www.xmswiki.com/xms/index.php?oldid=38609 *Contributors*: Ksmith, Unjedai

Strike Dip Markov Chain *Source*: http://www.xmswiki.com/xms/index.php?oldid=39782 *Contributors*: Jcreer, Ksmith

TSIM *Source*: http://www.xmswiki.com/xms/index.php?oldid=39783 *Contributors*: Jcreer, Ksmith, Unjedai

UTEXAS *Source*: http://www.xmswiki.com/xms/index.php?oldid=39699 *Contributors*: Jcreer, Ksmith, Unjedai

UTEXAS Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37224 *Contributors*: Ksmith, Tdye

WASH123D Display Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37225 *Contributors*: Jdye, Ksmith

Model Calibration *Source*: http://www.xmswiki.com/xms/index.php?oldid=35113 *Contributors*: Ksmith

Automated Parameter Estimation *Source*: http://www.xmswiki.com/xms/index.php?oldid=37909 *Contributors*: Alanlemon, Ksmith

Parameter Estimation Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=39685 *Contributors*: Jcreer, Ksmith

PEST *Source*: http://www.xmswiki.com/xms/index.php?oldid=46429 *Contributors*: Alanlemon, Ksmith

Run Options *Source*: http://www.xmswiki.com/xms/index.php?oldid=37117 *Contributors*: Ksmith

Observations *Source*: http://www.xmswiki.com/xms/index.php?oldid=37818 *Contributors*: Ksmith, Wood

Plot Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=39688 *Contributors*: Bdolinar, Jcreer, Ksmith

Calibration Targets *Source*: http://www.xmswiki.com/xms/index.php?oldid=37240 *Contributors*: Ksmith

Model Checker *Source*: http://www.xmswiki.com/xms/index.php?oldid=37241 *Contributors*: Ksmith, Wood

Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=39689 *Contributors*: Jcreer, Ksmith

Parameter Dialog *Source*: http://www.xmswiki.com/xms/index.php?oldid=37247 *Contributors*: Ksmith

Pilot Points *Source*: http://www.xmswiki.com/xms/index.php?oldid=37248 *Contributors*: Ksmith, Wood

Multiplier Arrays for Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=35129 *Contributors*: Ksmith

Standard MODFLOW Parameters *Source*: http://www.xmswiki.com/xms/index.php?oldid=39690 *Contributors*: Jcreer, Ksmith

Stochastic Modeling *Source*: http://www.xmswiki.com/xms/index.php?oldid=38615 *Contributors*: Ksmith

Gaussian Field Generator *Source*: http://www.xmswiki.com/xms/index.php?oldid=39741 *Contributors*: Jcreer, Ksmith

Risk Analysis Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=39694 *Contributors*: Jcreer, Ksmith

Importing Non-native GMS Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=37262 *Contributors*: Ksmith, Unjedai

Text Import Wizard *Source*: http://www.xmswiki.com/xms/index.php?oldid=39678 *Contributors*: Jcreer, Ksmith, Unjedai

Text Import Wizard Supported File Formats *Source*: http://www.xmswiki.com/xms/index.php?oldid=49449 *Contributors*: Alanlemon, Htran, Jcreer, Ksmith, Twood, Unjedai

Importing/Exporting CAD Data *Source*: http://www.xmswiki.com/xms/index.php?oldid=41262 *Contributors*: Ksmith, Unjedai

Importing USGS DEMs *Source*: http://www.xmswiki.com/xms/index.php?oldid=39680 *Contributors*: Jcreer, Ksmith, Unjedai

Importing/Exporting GIS Grids *Source*: http://www.xmswiki.com/xms/index.php?oldid=39681 *Contributors*: Jcreer, Ksmith, Unjedai

Importing/Exporting Shapefiles *Source*: http://www.xmswiki.com/xms/index.php?oldid=37273 *Contributors*: Ksmith, Unjedai

Shapefiles *Source*: http://www.xmswiki.com/xms/index.php?oldid=37274 *Contributors*: Ksmith

Import From Database *Source*: http://www.xmswiki.com/xms/index.php?oldid=37275 *Contributors*: Ksmith, Unjedai

Get Online Maps *Source*: http://www.xmswiki.com/xms/index.php?oldid=44953 *Contributors*: Cmsmemoe, Jcreer

Exporting Non-native GMS Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=39683 *Contributors*: Jcreer, Ksmith, Tdye, Unjedai

Images *Source*: http://www.xmswiki.com/xms/index.php?oldid=41111 *Contributors*: Jcreer, Ksmith, Unjedai

Registering an Image *Source*: http://www.xmswiki.com/xms/index.php?oldid=46623 *Contributors*: Jcreer, Ksmith

KMZ files *Source*: http://www.xmswiki.com/xms/index.php?oldid=39924 *Contributors*: Jcreer, Ksmith

GeoStudio Import *Source*: http://www.xmswiki.com/xms/index.php?oldid=33506 *Contributors*: Unjedai

File Formats *Source*: http://www.xmswiki.com/xms/index.php?oldid=33502 *Contributors*: Unjedai

Data Set Files *Source*: http://www.xmswiki.com/xms/index.php?oldid=39669 *Contributors*: Jcreer, Jdye, Ksmith

Bugfixes GMS *Source*: http://www.xmswiki.com/xms/index.php?oldid=48523 *Contributors*: Emccarthy, Glarsen, Htran, Jcreer, Ksmith, Unjedai

Dialog Help *Source*: http://www.xmswiki.com/xms/index.php?oldid=46337 *Contributors*: Alanlemon, Bdolinar, Jcreer, Ksmith, Unjedai

Image Sources, Licenses and Contributors

Image:Vertical_Displacement_b.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Vertical_Displacement_b.png *License*: unknown *Contributors*: Bdolinar
Image:fort_buchanan_b.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Fort_buchanan_b.png *License*: unknown *Contributors*: Unjedai
Image:geos3d_b.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Geos3d_b.png *License*: unknown *Contributors*: Unjedai
Image:Icon_TIN.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_TIN.gif *License*: unknown *Contributors*: Cballard
Image:Icon_BH.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_BH.gif *License*: unknown *Contributors*: Cballard
Image:Icon_Solid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_Solid.gif *License*: unknown *Contributors*: Cballard
Image:Icon_2Dmesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_2Dmesh.gif *License*: unknown *Contributors*: Cballard
Image:Icon_2Dgrid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_2Dgrid.gif *License*: unknown *Contributors*: Cballard
Image:Icon_2Dscat.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_2Dscat.gif *License*: unknown *Contributors*: Cballard
Image:Icon_3Dmesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_3Dmesh.gif *License*: unknown *Contributors*: Cballard
Image:Icon_3Dgrid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_3Dgrid.gif *License*: unknown *Contributors*: Cballard
Image:Icon_3Dscat.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_3Dscat.gif *License*: unknown *Contributors*: Cballard
Image:Icon_map.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_map.gif *License*: unknown *Contributors*: Cballard
Image:Icon_GIS.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Icon_GIS.gif *License*: unknown *Contributors*: Cballard
Image:GMS_screen.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:GMS_screen.jpg *License*: unknown *Contributors*: Ksmith
Image:editwind.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Editwind.jpg *License*: unknown *Contributors*: Ksmith
Image:graphics_window.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Graphics_window.jpg *License*: unknown *Contributors*: Ksmith
Image:Right_Click_GW.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Right_Click_GW.jpg *License*: unknown *Contributors*: Ksmith
Image:Tree_General.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_General.jpg *License*: unknown *Contributors*: Ksmith
Image:TimeStepWindow.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TimeStepWindow.jpg *License*: unknown *Contributors*: Ksmith
Image:StatusBar.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:StatusBar.jpg *License*: unknown *Contributors*: Ksmith
Image:ToolPalettes.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:ToolPalettes.jpg *License*: unknown *Contributors*: Ksmith
Image:macros.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Macros.gif *License*: unknown *Contributors*: Cballard
Image:static_tools.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Static_tools.png *License*: unknown *Contributors*: Unjedai
Image:Tool_Pan.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Pan.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Zoom.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Zoom.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Rotate.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Rotate.GIF *License*: unknown *Contributors*: Jhandy
Image:tool_measure.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_measure.png *License*: unknown *Contributors*: Unjedai
Image:3D_grid_tools.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:3D_grid_tools.gif *License*: unknown *Contributors*: Cballard
Image:MiniGridPlot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:MiniGridPlot.gif *License*: unknown *Contributors*: Cballard
Image:MiniGridToolbar.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:MiniGridToolbar.png *License*: unknown *Contributors*: Unjedai
Image:Modules_ToolBar.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Modules_ToolBar.gif *License*: unknown *Contributors*: Cballard
Image:dlg_register_gms.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dlg_register_gms.png *License*: unknown *Contributors*: Unjedai
Image:dlg_gms_ce_warning.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dlg_gms_ce_warning.png *License*: unknown *Contributors*: Unjedai
Image:gms_ce_watermark.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gms_ce_watermark.png *License*: unknown *Contributors*: Unjedai
Image:Material_Tools.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Material_Tools.png *License*: unknown *Contributors*: Unjedai
Image:material_set.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Material_set.png *License*: unknown *Contributors*: Unjedai
Image:material_set_in_project_explorer.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Material_set_in_project_explorer.png *License*: unknown *Contributors*: Unjedai
Image:ellipsiodcoordsys.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ellipsiodcoordsys.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys1.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys2.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys3.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cppcoordsys4.jpg *License*: unknown *Contributors*: Ksmith
Image:TransformDialogGMS.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:TransformDialogGMS.png *License*: unknown *Contributors*: Unjedai
Image:logcont.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Logcont.jpg *License*: unknown *Contributors*: Ksmith
Image:dispopts.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dispopts.gif *License*: unknown *Contributors*: Jdhowlett
Image:Display_TIN.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_TIN.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Borehole.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Borehole.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Solid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Solid.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_2DMesh.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_2DMesh.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_2DGrid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_2DGrid.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_2DScatterSet.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_2DScatterSet.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_3DMesh.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_3DMesh.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_3DGrid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_3DGrid.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_3DScatterSet.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_3DScatterSet.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Map.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Map.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_GIS.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_GIS.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_CrossSection.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_CrossSection.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Lighting.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Lighting.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_Axes.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_Axes.GIF *License*: unknown *Contributors*: Jhandy
Image:Display_DrawingGrid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Display_DrawingGrid.GIF *License*: unknown *Contributors*: Jhandy
Image:DrawingGrid.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:DrawingGrid.GIF *License*: unknown *Contributors*: Jhandy
Image:LightingDialog.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:LightingDialog.jpg *License*: unknown *Contributors*: Ksmith
Image:Axes.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Axes.GIF *License*: unknown *Contributors*: Jhandy
Image:AnnotationTree.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:AnnotationTree.jpg *License*: unknown *Contributors*: Djordan
Image:Scale.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Scale.jpg *License*: unknown *Contributors*: Sramiah
Image:North-arrow.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:North-arrow.jpg *License*: unknown *Contributors*: Ksmith
Image:Solids.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Solids.gif *License*: unknown *Contributors*: Cballard

Image:xsect.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect.png *License*: unknown *Contributors*: Unjedai
Image:Xsect_Sample.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect_Sample.png *License*: unknown *Contributors*: Unjedai
Image:Xsects_3dg.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsects_3dg.png *License*: unknown *Contributors*: Unjedai
Image:Xsect_3dm.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect_3dm.png *License*: unknown *Contributors*: Unjedai
Image:CrossSectionFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:CrossSectionFolder.GIF *License*: unknown *Contributors*: Jhandy
image:x2cov_1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:X2cov_1.png *License*: unknown *Contributors*: Unjedai
image:x2cov_2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:X2cov_2.png *License*: unknown *Contributors*: Unjedai
Image:SMS_Data_Calculator.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SMS_Data_Calculator.jpg *License*: unknown *Contributors*: Jcreer, Jdhowlett
Image:linear_eq1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Linear_eq1.jpg *License*: unknown *Contributors*: Ksmith
Image:convex_hull.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Convex_hull.jpg *License*: unknown *Contributors*: Ksmith
Image:Sample Interp.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sample_Interp.gif *License*: unknown *Contributors*: Jhandy
Image:shep_eq1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Shep_eq1.jpg *License*: unknown *Contributors*: Ksmith
Image:shep_eq2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Shep_eq2.jpg *License*: unknown *Contributors*: Ksmith
Image:shep_eq3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Shep_eq3.jpg *License*: unknown *Contributors*: Ksmith
Image:shep_eq4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Shep_eq4.jpg *License*: unknown *Contributors*: Ksmith
Image:shep_eq5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Shep_eq5.jpg *License*: unknown *Contributors*: Ksmith
Image:gradplanenode1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gradplanenode1.jpg *License*: unknown *Contributors*: Ksmith
Image:gradplanenode2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gradplanenode2.jpg *License*: unknown *Contributors*: Ksmith
Image:gradplanenode3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gradplanenode3.jpg *License*: unknown *Contributors*: Ksmith
Image:quadnodef1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Quadnodef1.jpg *License*: unknown *Contributors*: Ksmith
Image:quadnodef2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Quadnodef2.jpg *License*: unknown *Contributors*: Ksmith
Image:quadnodef3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Quadnodef3.jpg *License*: unknown *Contributors*: Ksmith
Image:quadnodef4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Quadnodef4.jpg *License*: unknown *Contributors*: Ksmith
Image:idw_fig1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Idw_fig1.gif *License*: unknown *Contributors*: Cballard
Image:idw_fig2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Idw_fig2.gif *License*: unknown *Contributors*: Cballard
Image:compinterp1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp1.jpg *License*: unknown *Contributors*: Ksmith
Image:idw_fig3.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Idw_fig3.gif *License*: unknown *Contributors*: Cballard
Image:compinterp2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp2.jpg *License*: unknown *Contributors*: Ksmith
Image:compinterp3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp3.jpg *License*: unknown *Contributors*: Ksmith
Image:compinterp4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp4.jpg *License*: unknown *Contributors*: Ksmith
Image:compinterp5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp5.jpg *License*: unknown *Contributors*: Ksmith
Image:compinterp6.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp6.jpg *License*: unknown *Contributors*: Ksmith
Image:compinterp7.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp7.jpg *License*: unknown *Contributors*: Ksmith
Image:compinterp8.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp8.jpg *License*: unknown *Contributors*: Ksmith
Image:compinterp9.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Compinterp9.jpg *License*: unknown *Contributors*: Ksmith
Image:Cttriangle1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cttriangle1.gif *License*: unknown *Contributors*: Jhandy
Image:cloughtocher1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Cloughtocher1.jpg *License*: unknown *Contributors*: Ksmith
Image:thiessen.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Thiessen.png *License*: unknown *Contributors*: Unjedai
Image:localcoord.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Localcoord.gif *License*: unknown *Contributors*: Cballard
Image:nnlambda.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Nnlambda.jpg *License*: unknown *Contributors*: Ksmith
Image:variogram.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogram.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq1.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq2.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq3.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq4.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq5.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq6.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq6.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq7.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq7.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq8.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq8.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq9.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq9.jpg *License*: unknown *Contributors*: Ksmith
Image:kriging eq10.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Kriging eq10.jpg *License*: unknown *Contributors*: Ksmith
Image:lags.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Lags.png *License*: unknown *Contributors*: Alanlemon
Image:variogrameq1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq1.jpg *License*: unknown *Contributors*: Ksmith
Image:tailhead.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tailhead.gif *License*: unknown *Contributors*: Cballard
Image:variogrameq2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq2.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq3.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq4.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq5.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq5a.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq5a.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq6.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq6.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq7.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq7.jpg *License*: unknown *Contributors*: Ksmith
Image:modelvar.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Modelvar.gif *License*: unknown *Contributors*: Cballard
Image:variogrameq8.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq8.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq9.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq9.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq10.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq10.jpg *License*: unknown *Contributors*: Ksmith
Image:variogrameq11.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Variogrameq11.jpg *License*: unknown *Contributors*: Ksmith
Image:azimuth.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Azimuth.gif *License*: unknown *Contributors*: Cballard
Image:anisotropy.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Anisotropy.gif *License*: unknown *Contributors*: Cballard
Image:Tins.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tins.gif *License*: unknown *Contributors*: Cballard
Image:tins.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tins.gif *License*: unknown *Contributors*: Cballard
Image:tinsub.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tinsub.gif *License*: unknown *Contributors*: Cballard

Image:breakline.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Breakline.gif *License*: unknown *Contributors*: Cballard
Image:TINFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:TINFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:contour_w_tri.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Contour_w_tri.gif *License*: unknown *Contributors*: Cballard
Image:contour_wo_tri.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Contour_wo_tri.gif *License*: unknown *Contributors*: Cballard
Image:thiessen.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Thiessen.gif *License*: unknown *Contributors*: Cballard
Image:circum.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Circum.gif *License*: unknown *Contributors*: Cballard
Image:Tool_SelectVertices.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectVertices.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectTriangles.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectTriangles.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectTINs.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectTINs.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectVertexString.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectVertexString.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateVertices.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateVertices.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateTriangles.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateTriangles.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SwapEdges.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SwapEdges.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_ContourLabels.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_ContourLabels.GIF *License*: unknown *Contributors*: Jhandy
Image:stratigmod.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stratigmod.gif *License*: unknown *Contributors*: Cballard
Image:del_criterion.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Del_criterion.gif *License*: unknown *Contributors*: Cballard
Image:Holes1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Holes1.gif *License*: unknown *Contributors*: Jhandy
Image:Sampledata1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sampledata1.gif *License*: unknown *Contributors*: Jhandy
Image:Hydrogeologic_Unit_IDs.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hydrogeologic_Unit_IDs.gif *License*: unknown *Contributors*: Cballard
Image:BoreholeFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:BoreholeFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectBorhole.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectBorhole.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectSegment.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectSegment.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectContact.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectContact.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectCrossSection.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectCrossSection.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateBorehole.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateBorehole.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateContact.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateContact.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateCrossSection.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateCrossSection.GIF *License*: unknown *Contributors*: Jhandy
Image:xsects.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsects.gif *License*: unknown *Contributors*: Cballard
Image:Hydrogeologic_Unit_Display.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hydrogeologic_Unit_Display.gif *License*: unknown *Contributors*: Cballard
Image:Xsect_Blank.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect_Blank.png *License*: unknown *Contributors*: Unjedai
Image:Tool_Select.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Select.gif *License*: unknown *Contributors*: Cballard
Image:Tool_SelectNode.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectNode.gif *License*: unknown *Contributors*: Cballard
Image:Tool_SelectVertex.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectVertex.gif *License*: unknown *Contributors*: Cballard
Image:Tool_SelectArc.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectArc.gif *License*: unknown *Contributors*: Cballard
Image:Tool_SelectPoly.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectPoly.gif *License*: unknown *Contributors*: Cballard
Image:Tool_CreateVertex.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateVertex.gif *License*: unknown *Contributors*: Cballard
Image:Tool_CreateArc.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateArc.gif *License*: unknown *Contributors*: Cballard
Image:Tool_FrameAllXSects.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_FrameAllXSects.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_FrameCurrentXSect.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_FrameCurrentXSect.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_ZMagnification.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_ZMagnification.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_PlotOptions.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_PlotOptions.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Delete.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Delete.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Left.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Left.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Right.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Right.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Print.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Print.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_AutoMatch.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_AutoMatch.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreatePoly.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreatePoly.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_DeleteAll.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_DeleteAll.GIF *License*: unknown *Contributors*: Jhandy
Image:samp_bor.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_bor.gif *License*: unknown *Contributors*: Cballard
Image:Hor_Asg_TIN.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hor_Asg_TIN.gif *License*: unknown *Contributors*: Cballard
Image:HorCoverage.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HorCoverage.png *License*: unknown *Contributors*: Cballard
Image:HorCoverageSolid.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:HorCoverageSolid.png *License*: unknown *Contributors*: Cballard
Image:SlopeXsectBore.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsectBore.png *License*: unknown *Contributors*: Alanlemon
Image:SlopeXsectTop.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsectTop.png *License*: unknown *Contributors*: Alanlemon
Image:SlopeXsectFailure.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsectFailure.png *License*: unknown *Contributors*: Alanlemon
Image:SlopeXsect.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsect.png *License*: unknown *Contributors*: Alanlemon
Image:SlopeXsectUserXsect.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsectUserXsect.png *License*: unknown *Contributors*: Alanlemon
Image:SlopeXsectZoom.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsectZoom.png *License*: unknown *Contributors*: Alanlemon
Image:SlopeXsectSolids.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsectSolids.png *License*: unknown *Contributors*: Alanlemon
Image:SlopeXsectSolidsXSects.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsectSolidsXSects.png *License*: unknown *Contributors*: Alanlemon
Image:SlopeXsectSolidsXSectsZoom.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:SlopeXsectSolidsXSectsZoom.png *License*: unknown *Contributors*: Alanlemon
Image:Xsect_hor_application.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect_hor_application.png *License*: unknown *Contributors*: Alanlemon
Image:Tin1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tin1.png *License*: unknown *Contributors*: Alanlemon
Image:Tin2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tin2.png *License*: unknown *Contributors*: Alanlemon
Image:Solids_river.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Solids_river.png *License*: unknown *Contributors*: Alanlemon
Image:Solids_StudyArea.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Solids_StudyArea.png *License*: unknown *Contributors*: Alanlemon
Image:AllSolids.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:AllSolids.png *License*: unknown *Contributors*: Alanlemon
Image:AllSolids_XsectXY.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:AllSolids_XsectXY.png *License*: unknown *Contributors*: Alanlemon
Image:AllSolids_Xsect.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:AllSolids_Xsect.png *License*: unknown *Contributors*: Alanlemon
Image:samp_huf.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_huf.gif *License*: unknown *Contributors*: Cballard
Image:samp_solids.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_solids.gif *License*: unknown *Contributors*: Cballard

Image:samp_xsect.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_xsect.gif *License*: unknown *Contributors*: Cballard
Image:Hor_TINs.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hor_TINs.gif *License*: unknown *Contributors*: Cballard
Image:Hor_3DMesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Hor_3DMesh.gif *License*: unknown *Contributors*: Cballard
Image:solids.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Solids.gif *License*: unknown *Contributors*: Cballard
Image:xsect.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Xsect.gif *License*: unknown *Contributors*: Cballard
Image:SolidsFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:SolidsFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectSolids.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectSolids.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectFace.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectFace.GIF *License*: unknown *Contributors*: Jhandy
Image:meshfromsolids.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Meshfromsolids.gif *License*: unknown *Contributors*: Cballard
Image:huf.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Huf.gif *License*: unknown *Contributors*: Cballard
Image:grid1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Grid1.gif *License*: unknown *Contributors*: Cballard
Image:layer_range1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Layer_range1.gif *License*: unknown *Contributors*: Cballard
Image:layer_range2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Layer_range2.gif *License*: unknown *Contributors*: Cballard
Image:grid2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Grid2.gif *License*: unknown *Contributors*: Cballard
Image:gridx2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gridx2.gif *License*: unknown *Contributors*: Cballard
Image:gridx3.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gridx3.gif *License*: unknown *Contributors*: Cballard
Image:small_min_thick.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Small_min_thick.jpg *License*: unknown *Contributors*: Cballard
Image:large_min_thick.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Large_min_thick.jpg *License*: unknown *Contributors*: Cballard
Image:overlaygrid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Overlaygrid.gif *License*: unknown *Contributors*: Cballard
Image:overlayxsect2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Overlayxsect2.gif *License*: unknown *Contributors*: Cballard
Image:overlayxsect1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Overlayxsect1.gif *License*: unknown *Contributors*: Cballard
Image:solidstomodflow1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Solidstomodflow1.jpg *License*: unknown *Contributors*: Ksmith
Image:solidstomodflow2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Solidstomodflow2.jpg *License*: unknown *Contributors*: Ksmith
Image:keq.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Keq.gif *License*: unknown *Contributors*: Cballard
Image:2d mesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:2d_mesh.gif *License*: unknown *Contributors*: Cballard
Image:2d_to_3d_mesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:2d_to_3d_mesh.gif *License*: unknown *Contributors*: Cballard
Image:samp_seep.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_seep.gif *License*: unknown *Contributors*: Cballard
Image:2DMeshFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:2DMeshFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateLinTriElement.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateLinTriElement.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateQuadLinElement.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateQuadLinElement.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateLinQuadElement.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateLinQuadElement.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreateQuadQuadElement.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreateQuadQuadElement.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_MergeSplit.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_MergeSplit.GIF *License*: unknown *Contributors*: Jhandy
Image:Nodes1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Nodes1.gif *License*: unknown *Contributors*: Jhandy
Image:top_bot_mesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Top_bot_mesh.gif *License*: unknown *Contributors*: Cballard
Image:tet_mesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tet_mesh.gif *License*: unknown *Contributors*: Cballard
Image:2d_mesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:2d_mesh.gif *License*: unknown *Contributors*: Cballard
Image:var_tet_mesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Var_tet_mesh.gif *License*: unknown *Contributors*: Cballard
Image:elem_1d_linear.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_1d_linear.png *License*: unknown *Contributors*: Unjedai
Image:elem_1d_quad.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_1d_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_transition.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_transition.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_linear_tri.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_linear_tri.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_tri.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_quad_tri.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_linear_quad.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_linear_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_quad.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_quad_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_quad_wcenter.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_2d_quad_quad_wcenter.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_tet.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_3d_linear_tet.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_prism.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_3d_linear_prism.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_hex.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_3d_linear_hex.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_pyramid.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Elem_3d_linear_pyramid.png *License*: unknown *Contributors*: Unjedai
Image:adapt.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Adapt.gif *License*: unknown *Contributors*: Cballard
Image:paving.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Paving.gif *License*: unknown *Contributors*: Cballard
Image:degen_edge.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Degen_edge.gif *License*: unknown *Contributors*: Cballard
Image:Grid interp.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Grid_interp.gif *License*: unknown *Contributors*: Cballard
Image:grid_interp.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Grid_interp.gif *License*: unknown *Contributors*: Cballard
Image:Sample_water_table.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sample_water_table.GIF *License*: unknown *Contributors*: Jhandy
Image:GridType1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:GridType1.gif *License*: unknown *Contributors*: Jhandy
Image:2DGridFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:2DGridFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectCell.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectCell.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectI.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectI.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectJ.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectJ.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectNode.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectNode.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_AddIBoundary.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_AddIBoundary.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_AddJBoundary.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_AddJBoundary.GIF *License*: unknown *Contributors*: Jhandy
Image:Samp 2dscat.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_2dscat.gif *License*: unknown *Contributors*: Cballard
Image:Sample_LayInterp.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sample_LayInterp.gif *License*: unknown *Contributors*: Jhandy
Image:samp_2dscat.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_2dscat.gif *License*: unknown *Contributors*: Cballard
Image:2DScatterFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:2DScatterFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectScatterSet.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectScatterSet.GIF *License*: unknown *Contributors*: Jhandy
Image:gaussian_sim_options.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gaussian_sim_options.gif *License*: unknown *Contributors*: Cballard
Image:gaussian_fieldgen_wrapper.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gaussian_fieldgen_wrapper.gif *License*: unknown *Contributors*: Cballard

Image:gaussian_sim_results.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gaussian_sim_results.gif *License*: unknown *Contributors*: Cballard
Image:image06.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image06.gif *License*: unknown *Contributors*: Cballard
Image:Sample FEMWATER.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sample_FEMWATER.JPG *License*: unknown *Contributors*: Jhandy
Image:Sample_FEMWATER.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sample_FEMWATER.JPG *License*: unknown *Contributors*: Jhandy
Image:ElemTypes1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:ElemTypes1.gif *License*: unknown *Contributors*: Jhandy
Image:zones1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Zones1.gif *License*: unknown *Contributors*: Jhandy
Image:Extrusion1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Extrusion1.gif *License*: unknown *Contributors*: Jhandy
Image:Seam1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Seam1.gif *License*: unknown *Contributors*: Jhandy
Image:Refine1a.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Refine1a.gif *License*: unknown *Contributors*: Jhandy
Image:Refine2b.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Refine2b.GIF *License*: unknown *Contributors*: Jhandy
Image:Refine3a.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Refine3a.gif *License*: unknown *Contributors*: Jhandy
Image:Refine4b.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Refine4b.gif *License*: unknown *Contributors*: Jhandy
Image:3DMeshFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:3DMeshFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectBoundaryNodes.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectBoundaryNodes.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectBoundaryFaces.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectBoundaryFaces.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectMaterialZones.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectMaterialZones.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectElements.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectElements.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectNodes.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectNodes.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectNodeStrings.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectNodeStrings.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectWells.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectWells.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectParticleStartingLoc.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectParticleStartingLoc.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_DefineTetrahedron.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_DefineTetrahedron.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_DefinePyramidElement.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_DefinePyramidElement.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_DefineWedgeElement.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_DefineWedgeElement.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_DefineHexahedronElement.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_DefineHexahedronElement.GIF *License*: unknown *Contributors*: Jhandy
Image:isosurf_ss.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Isosurf_ss.jpg *License*: unknown *Contributors*: Ksmith
Image:isosurface.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Isosurface.gif *License*: unknown *Contributors*: Cballard
Image:iso_ss.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Iso_ss.jpg *License*: unknown *Contributors*: Ksmith
Image:iso_cap.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Iso_cap.jpg *License*: unknown *Contributors*: Ksmith
Image:meshgenc.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Meshgenc.gif *License*: unknown *Contributors*: Cballard
Image:samp_mt3d.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_mt3d.gif *License*: unknown *Contributors*: Cballard
Image:GridType2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:GridType2.gif *License*: unknown *Contributors*: Jhandy
Image:Tool_AddIRow.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_AddIRow.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_AddJRow.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_AddJRow.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_AddKRow.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_AddKRow.GIF *License*: unknown *Contributors*: Jhandy
Image:3DGridFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:3DGridFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectCells.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectCells.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectGridNode.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectGridNode.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectIRow.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectIRow.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectJRow.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectJRow.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectKRow.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectKRow.GIF *License*: unknown *Contributors*: Jhandy
Image:named_layer_ranges.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Named_layer_ranges.png *License*: unknown *Contributors*: Unjedai
Image:Sample_Interp.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sample_Interp.gif *License*: unknown *Contributors*: Jhandy
Image:3DScatterFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:3DScatterFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Easttex.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Easttex.jpg *License*: unknown *Contributors*: Cballard
Image:easttex.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Easttex.jpg *License*: unknown *Contributors*: Cballard
Image:featureobjects.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Featureobjects.gif *License*: unknown *Contributors*: Cballard
Image:redverts.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Redverts.gif *License*: unknown *Contributors*: Cballard
Image:polygons.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Polygons.gif *License*: unknown *Contributors*: Cballard
Image:MapFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:MapFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectArcGroup.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectArcGroup.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_GridFrame.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_GridFrame.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_CreatePoint.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_CreatePoint.GIF *License*: unknown *Contributors*: Jhandy
Image:Tree_Map.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_Map.gif *License*: unknown *Contributors*: Cballard
Image:grid_frame_tool.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Grid_frame_tool.gif *License*: unknown *Contributors*: Cballard
Image:Discretization01.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Discretization01.jpg *License*: unknown *Contributors*: Ksmith
Image:dam_xsect.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dam_xsect.gif *License*: unknown *Contributors*: Cballard
Image:dam_mesh.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Dam_mesh.gif *License*: unknown *Contributors*: Cballard
Image:GISFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:GISFolder.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectFeatures.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectFeatures.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Identify.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_Identify.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_SelectGISElements.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_SelectGISElements.GIF *License*: unknown *Contributors*: Jhandy
Image:Topo.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Topo.gif *License*: unknown *Contributors*: Cballard
Image:samp_map.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Samp_map.gif *License*: unknown *Contributors*: Cballard
Image:screenlogic.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Screenlogic.gif *License*: unknown *Contributors*: Cballard
Image:time.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Time.gif *License*: unknown *Contributors*: Cballard
Image:Switching_MF2K5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Switching_MF2K5.jpg *License*: unknown *Contributors*: Bdolinar, Htran
Image:MF2KLPF.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:MF2KLPF.jpg *License*: unknown *Contributors*: Htran
Image:MF2K5LPF.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:MF2K5LPF.jpg *License*: unknown *Contributors*: Htran
Image:changinghead.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Changinghead.gif *License*: unknown *Contributors*: Cballard
Image:drt.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Drt.png *License*: unknown *Contributors*: Unjedai

Image:Yes_check.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Yes_check.png *License*: unknown *Contributors*: Unjedai
Image:ofr00-466_1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ofr00-466_1.png *License*: unknown *Contributors*: Unjedai
Image:ets_xy_1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ets_xy_1.png *License*: unknown *Contributors*: Unjedai
Image:ets_xy_2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ets_xy_2.png *License*: unknown *Contributors*: Unjedai
Image:evt.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Evt.png *License*: unknown *Contributors*: Unjedai
Image:flowbarrier.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Flowbarrier.gif *License*: unknown *Contributors*: Cballard
Image:huf_grid.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Huf_grid.gif *License*: unknown *Contributors*: Cballard
Image:huf_grid_fill.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Huf_grid_fill.gif *License*: unknown *Contributors*: Cballard
Image:mnw.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mnw.png *License*: unknown *Contributors*: Unjedai
Image:mnw2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Mnw2.png *License*: unknown *Contributors*: Bdolinar
Image:stream.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stream.gif *License*: unknown *Contributors*: Cballard
Image:sitemap.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Sitemap.gif *License*: unknown *Contributors*: Cballard
Image:concmod.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Concmod.gif *License*: unknown *Contributors*: Cballard
Image:gridmod.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gridmod.gif *License*: unknown *Contributors*: Cballard
Image:Activate1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Activate1.gif *License*: unknown *Contributors*: Jhandy
Image:tool_insert_row.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_insert_row.gif *License*: unknown *Contributors*: Cballard
Image:tool_remove_row.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tool_remove_row.gif *License*: unknown *Contributors*: Cballard
Image:maptomodflow1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Maptomodflow1.jpg *License*: unknown *Contributors*: Ksmith
Image:well_flux_conv.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Well_flux_conv.gif *License*: unknown *Contributors*: Cballard
Image:maptomodflow2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Maptomodflow2.jpg *License*: unknown *Contributors*: Ksmith
File:Map-layer-range.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Map-layer-range.png *License*: unknown *Contributors*: Bdolinar
File:Map-single-cell.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Map-single-cell.png *License*: unknown *Contributors*: Bdolinar
File:Map-including-lower-cells.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Map-including-lower-cells.png *License*: unknown *Contributors*: Bdolinar
Image:layerele1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Layerele1.jpg *License*: unknown *Contributors*: Ksmith
Image:layerele2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Layerele2.jpg *License*: unknown *Contributors*: Ksmith
Image:layerele3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Layerele3.jpg *License*: unknown *Contributors*: Ksmith
Image:layerele4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Layerele4.jpg *License*: unknown *Contributors*: Ksmith
Image:layerele5.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Layerele5.jpg *License*: unknown *Contributors*: Ksmith
Image:ParametersDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ParametersDialog.png *License*: unknown *Contributors*: Bdolinar
Image:paramInst.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ParamInst.png *License*: unknown *Contributors*: Unjedai
Image:eq_cond1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Eq_cond1.gif *License*: unknown *Contributors*: Ksmith, Kwinters
Image:eq_cond2.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Eq_cond2.gif *License*: unknown *Contributors*: Ksmith, Kwinters
Image:eq_cond3.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Eq_cond3.gif *License*: unknown *Contributors*: Ksmith, Kwinters
Image:eq_cond4.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Eq_cond4.gif *License*: unknown *Contributors*: Ksmith, Kwinters
Image:eq_cond5.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Eq_cond5.gif *License*: unknown *Contributors*: Ksmith, Kwinters
Image:eq_cond6.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Eq_cond6.gif *License*: unknown *Contributors*: Ksmith, Kwinters
Image:eq_cond7.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Eq_cond7.gif *License*: unknown *Contributors*: Ksmith, Kwinters
File:Rch.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Rch.PNG *License*: unknown *Contributors*: Htran
File:Chd.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Chd.PNG *License*: unknown *Contributors*: Htran
File:GHB.PNG *Source*: http://www.xmswiki.com/xms/index.php?title=File:GHB.PNG *License*: unknown *Contributors*: Alanlemon
Image:MODFLOWFolder.GIF *Source*: http://www.xmswiki.com/xms/index.php?title=File:MODFLOWFolder.GIF *License*: unknown *Contributors*: Jhandy
image:modflowpostviewing1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Modflowpostviewing1.jpg *License*: unknown *Contributors*: Ksmith
image:modflowpostviewing2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Modflowpostviewing2.jpg *License*: unknown *Contributors*: Ksmith
image:modflowpostviewing3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Modflowpostviewing3.jpg *License*: unknown *Contributors*: Ksmith
Image:ModflowProperties.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:ModflowProperties.JPG *License*: unknown *Contributors*: Jdye
Image:ModflowPropertiesDialogBox.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:ModflowPropertiesDialogBox.jpg *License*: unknown *Contributors*: Alanlemon
Image:ErrorSummary.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:ErrorSummary.gif *License*: unknown *Contributors*: Abarney
Image:zone_budget.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Zone_budget.gif *License*: unknown *Contributors*: Cballard
Image:Flow plot.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Flow_plot.jpg *License*: unknown *Contributors*: Cballard
Image:Plot values.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Plot_values.JPG *License*: unknown *Contributors*: Cballard, Unjedai
Image:CCF_datasets.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:CCF_datasets.gif *License*: unknown *Contributors*: Cballard
Image:modpath_pathline_report.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Modpath_pathline_report.gif *License*: unknown *Contributors*: Cballard
Image:modpath_export_file.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Modpath_export_file.gif *License*: unknown *Contributors*: Cballard
Image:flownet1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Flownet1.gif *License*: unknown *Contributors*: Jhandy
Image:SEEP2D_CM.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:SEEP2D_CM.gif *License*: unknown *Contributors*: Cballard
Image:seep2ddisplay1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Seep2ddisplay1.jpg *License*: unknown *Contributors*: Ksmith
Image:Matset.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Matset.gif *License*: unknown *Contributors*: Cballard
Image:pathlines.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Pathlines.png *License*: unknown *Contributors*: Unjedai
Image:markov_graphs.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Markov_graphs.png *License*: unknown *Contributors*: Unjedai
image:tprogseq1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tprogseq1.jpg *License*: unknown *Contributors*: Ksmith
image:tprogseq2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tprogseq2.jpg *License*: unknown *Contributors*: Ksmith
Image:rate_matrix.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Rate_matrix.gif *License*: unknown *Contributors*: Cballard
Image:t-progs_wo_boreholes.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:T-progs_wo_boreholes.png *License*: unknown *Contributors*: Unjedai
Image:t-progs_w_boreholes.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:T-progs_w_boreholes.png *License*: unknown *Contributors*: Unjedai
Image:azimuth1.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Azimuth1.png *License*: unknown *Contributors*: Unjedai
Image:azimuth2.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Azimuth2.png *License*: unknown *Contributors*: Unjedai
Image:matset_samp.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Matset_samp.png *License*: unknown *Contributors*: Unjedai
Image:huf_sample.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Huf_sample.gif *License*: unknown *Contributors*: Cballard
Image:markov_chains_z.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Markov_chains_z.png *License*: unknown *Contributors*: Unjedai
Image:markov_chain_plot.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Markov_chain_plot.png *License*: unknown *Contributors*: Unjedai
Image:gameas.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Gameas.png *License*: unknown *Contributors*: Unjedai

Image:verticalchain2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Verticalchain2.jpg *License*: unknown *Contributors*: Ksmith
Image:verticalchain3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Verticalchain3.jpg *License*: unknown *Contributors*: Ksmith
Image:markov_chains_x.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:Markov_chains_x.png *License*: unknown *Contributors*: Unjedai
Image:strikedipchain1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Strikedipchain1.jpg *License*: unknown *Contributors*: Ksmith
Image:UtexasGmsScreenShot.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:UtexasGmsScreenShot.png *License*: unknown *Contributors*: Unjedai
Image:ProfileLinesExampleB.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ProfileLinesExampleB.png *License*: unknown *Contributors*: Unjedai
Image:ProfileLinesExampleA.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:ProfileLinesExampleA.png *License*: unknown *Contributors*: Unjedai
Image:UTEXASTreeModelIcon.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:UTEXASTreeModelIcon.png *License*: unknown *Contributors*: Unjedai
Image:UtexasHeadDisplayOption.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:UtexasHeadDisplayOption.png *License*: unknown *Contributors*: Unjedai
Image:calib_plot.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Calib_plot.jpg *License*: unknown *Contributors*: Cballard
Image:observeq1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Observeq1.jpg *License*: unknown *Contributors*: Ksmith
Image:observeq2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Observeq2.jpg *License*: unknown *Contributors*: Ksmith
Image:comp_v_obs_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Comp_v_obs_plot.gif *License*: unknown *Contributors*: Cballard
Image:comp_v_obs_wt_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Comp_v_obs_wt_plot.gif *License*: unknown *Contributors*: Cballard
Image:res_v_obs_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Res_v_obs_plot.gif *License*: unknown *Contributors*: Cballard
Image:res_v_obs_wt_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Res_v_obs_wt_plot.gif *License*: unknown *Contributors*: Cballard
Image:par_sen_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Par_sen_plot.gif *License*: unknown *Contributors*: Cballard
Image:error_v_sim_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Error_v_sim_plot.gif *License*: unknown *Contributors*: Cballard
Image:error_v_time_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Error_v_time_plot.gif *License*: unknown *Contributors*: Cballard
Image:err_summary_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Err_summary_plot.gif *License*: unknown *Contributors*: Cballard
Image:time_series_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Time_series_plot.gif *License*: unknown *Contributors*: Cballard
Image:active_data_time_series_plot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Active_data_time_series_plot.gif *License*: unknown *Contributors*: Cballard
Image:flowbudgetplot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Flowbudgetplot.gif *License*: unknown *Contributors*: Cballard
Image:Calibtarget1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Calibtarget1.gif *License*: unknown *Contributors*: Jhandy
Image:zonation.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Zonation.gif *License*: unknown *Contributors*: Cballard
Image:pilot.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Pilot.gif *License*: unknown *Contributors*: Cballard
Image:Pilot_Point_Cond.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Pilot_Point_Cond.gif *License*: unknown *Contributors*: Cballard
Image:multiplier_array_param_dialog.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Multiplier_array_param_dialog.gif *License*: unknown *Contributors*: Cballard
Image:stochasticeq1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stochasticeq1.jpg *License*: unknown *Contributors*: Ksmith
Image:stochasticeq2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stochasticeq2.jpg *License*: unknown *Contributors*: Ksmith
Image:freq.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Freq.gif *License*: unknown *Contributors*: Cballard
Image:stochasticeq3.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stochasticeq3.jpg *License*: unknown *Contributors*: Ksmith
Image:stochasticeq4.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Stochasticeq4.jpg *License*: unknown *Contributors*: Ksmith
Image:StatAnalysis.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:StatAnalysis.gif *License*: unknown *Contributors*: Kwinters
Image:Tree_Risk_Anaylsis.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Tree_Risk_Anaylsis.gif *License*: unknown *Contributors*: Cballard
Image:ra_3.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ra_3.gif *License*: unknown *Contributors*: Cballard
Image:ra_4.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ra_4.gif *License*: unknown *Contributors*: Cballard
Image:riskwizardeq1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Riskwizardeq1.jpg *License*: unknown *Contributors*: Ksmith
Image:alpha_char.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Alpha_char.gif *License*: unknown *Contributors*: Cballard
Image:riskwizardeq2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Riskwizardeq2.jpg *License*: unknown *Contributors*: Ksmith
Image:wt_plot1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Wt_plot1.gif *License*: unknown *Contributors*: Jhandy
Image:ra_1.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Ra_1.jpg *License*: unknown *Contributors*: Cballard
Image:import_wiz_page1.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Import_wiz_page1.gif *License*: unknown *Contributors*: Cballard
Image:TI_2.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:TI_2.jpg *License*: unknown *Contributors*: Cballard
Image:image166.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:Image166.jpg *License*: unknown *Contributors*: Cballard
File:GetOnlineMapsDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GetOnlineMapsDialog.png *License*: unknown *Contributors*: Cmsmemoe
Image:texturemap.gif *Source*: http://www.xmswiki.com/xms/index.php?title=File:Texturemap.gif *License*: unknown *Contributors*: Cballard
Image:SelectRegistrationPoint.jpg *Source*: http://www.xmswiki.com/xms/index.php?title=File:SelectRegistrationPoint.jpg *License*: unknown *Contributors*: Abarney
Image:Zoom.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Zoom.JPG *License*: unknown *Contributors*: Jdhowlett
Image:Pan.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Pan.JPG *License*: unknown *Contributors*: Jdhowlett
Image:Frame.JPG *Source*: http://www.xmswiki.com/xms/index.php?title=File:Frame.JPG *License*: unknown *Contributors*: Jdhowlett
Image:GoogleEarth.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GoogleEarth.png *License*: unknown *Contributors*: Unjedai
Image:GeoStudioImportDialog.png *Source*: http://www.xmswiki.com/xms/index.php?title=File:GeoStudioImportDialog.png *License*: unknown *Contributors*: Djordan